Instruction Report ITL-94-1
April 1994

US Army Corps of Engineers
Waterways Experiment Station

AD-A280 147

Computer-Aided Structural Engineering (CASE) Project

Tutorial Guide: Computer-Aided Structural Modeling (CASM)

Version 5.00

by *David Wickersheimer, Gene McDermott, Ken Taylor, Carl Roth*
Wickersheimer Engineers, Inc.

WES

Approved For Public Release; Distribution Is Unlimited

Prepared for Headquarters, U.S. Army Corps of Engineers

Computer-Aided Structural Engineering (CASE) Project

Instruction Report ITL-94-1
April 1994

Tutorial Guide: Computer-Aided Structural Modeling (CASM)

Version 5.00

by David Wickersheimer, Gene McDermott, Ken Taylor, Carl Roth

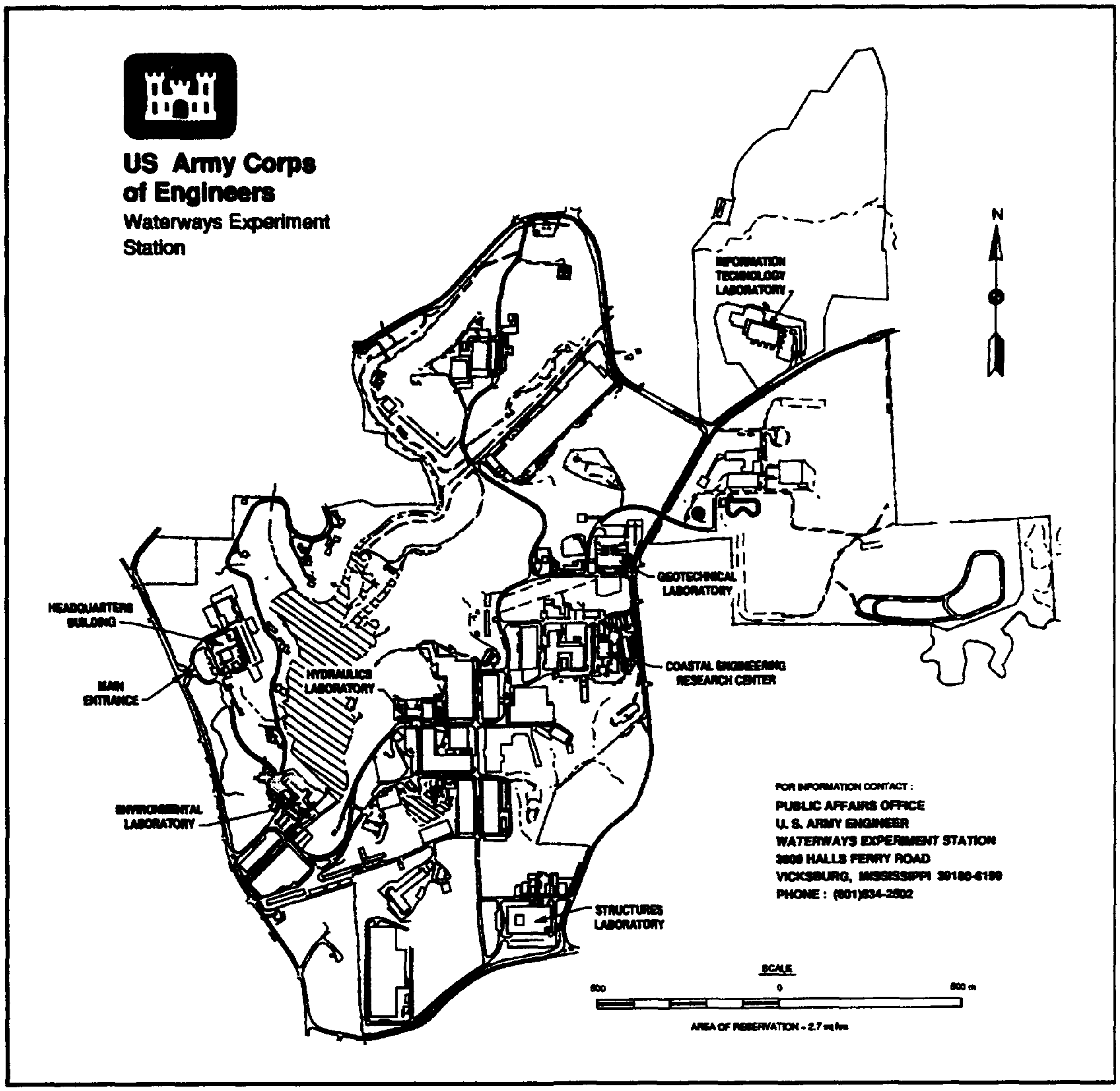
Wickersheimer Engineers, Inc.
821 South Neil St.
Champaign, IL 61820

Final report

Approved for public release; distribution is unlimited

Prepared for U.S. Army Corps of Engineers
Washington, DC 20314-1000

Monitored by Information Technology Laboratory
U.S. Army Engineer Waterways Experiment Station
3909 Halls Ferry Road, Vicksburg, MS 39180-6199



Waterways Experiment Station Cataloging-in-Publication Data

Tutorial guide : Computer-Aided Structural Modeling (CASM), version 5.00 / by David Wickersheimer ... [et al.] ; prepared for U.S. Army Corps of Engineers, monitored by Information Technology Laboratory, U.S. Army Engineer Waterways Experiment Station.

303 p. ; 28 cm. -- (Instruction report ; ITL-94-1)

Includes index.

1. Computer-aided design -- Handbooks, manuals, etc. 2. Structural engineering -- Computer programs. 3. CASM (Computer program. I. Wickersheimer, David. II. Computer-Aided Structural Engineering (CASE) Project. III. Information Technology Laboratory (US Army Corps of Engineers, Waterways Experiment Station) IV. U.S. Army Engineer Waterways Experiment Station. V. Instruction report (U.S. Army Engineer Waterways Experiment Station) ; ITL-94-1.

TA7 W34i no.ITL-94-1

PREFACE

This tutorial describes the use of the computer program CASM, which is designed to aid the structural engineer in the preliminary design and evaluation of structural building systems by the use of three-dimensional interactive graphics. Funds for the development of this program and publication of this report were provided to the Information Technology Laboratory (ITL), U.S. Army Engineer Waterways Experiment Station (WES), Vicksburg, MS, by the Directorate of Military Programs, Headquarters, U.S. Army Corps of Engineers (HQUSACE), under the Research, Development, Test, and Evaluation (RDT&E) program. The work was accomplished under Work Unit No. AT40-CA-001 entitled "CASE (Computer Aided Structural Engineering) Building Systems." The work was performed by members of Wickersheimer Engineers, Inc., of Champaign, IL, under Contract No. DACA39-86-C-0024.

Specifications for the program were provided by members of the Building Systems Task Group of the CASE Project. The following were members of the task group during program development:

Mr. Dan Reynolds, U.S. Army Engineer (USAE) District, Sacramento (Chairman)

Ms. Anjana Chudgar, USAE Division, Ohio River

Mr. Pete Rossbach, USAE District, Baltimore

Mr. Dave Smith, USAE District, Omaha

Mr. Mark Burkholder, USAE District, Tulsa

Mr. Jerry Maurseth, USAE District, Portland

Mr. Chris Merrill, WES

Mr. Michael Pace, WES

The computer program and tutorial were written by Messrs. David Wickersheimer, Gene McDermott, Ken Taylor, and Carl Roth of Wickersheimer Engineers, Inc.

The work was monitored at WES by Mr. Michael E. Pace and Mr. Chris Merrill, Computer-Aided Engineering Division (CAED), under the general supervision of Mr. H. Wayne Jones, Chief, Scientific and Engineering Applications Center; Dr. Reed Mosher, Chief, CAED; Mr. Timothy Ables, Assistant Director, ITL; and Dr. N. Radhakrishnan, Director, ITL. Mr. Donald Dressler was the original HQUSACE point of contact, and Mr. Charles Gutberlet is the present technical monitor.

Dr. Robert W. Whalin is Director of WES. COL Bruce K. Howard, EN, is Commander.

CONTENTS

INTRODUCTION

This tutorial guide will take you through a series of example design problems step by step to acquaint you with all applications of the program. The basic reference for the example problems is found in the Appendices of Load Assumptions for Buildings, TM 5-809-1/ AFM 88-3, Chapter 1 - Technical Manual, 1986 edition. This document in general adopts the A.N.S.I. A58.1-1982, Minimum Design Loads for Buildings and Other Structures, which has been nationally adopted in various forms by the model building codes, such as BOCA.

It is assumed that you have completed the INSTALLATION and PROGRAM OVERVIEW chapters in the CASM Guide and have the CASM program window displayed on the monitor. Refer also to the REFERENCE chapter in the CASM User's Guide, which contains detailed steps and illustrations for all the CASM commands. You are encouraged to begin each application of CASM by inputting design criteria that are consistently needed by the program to calculate load data.

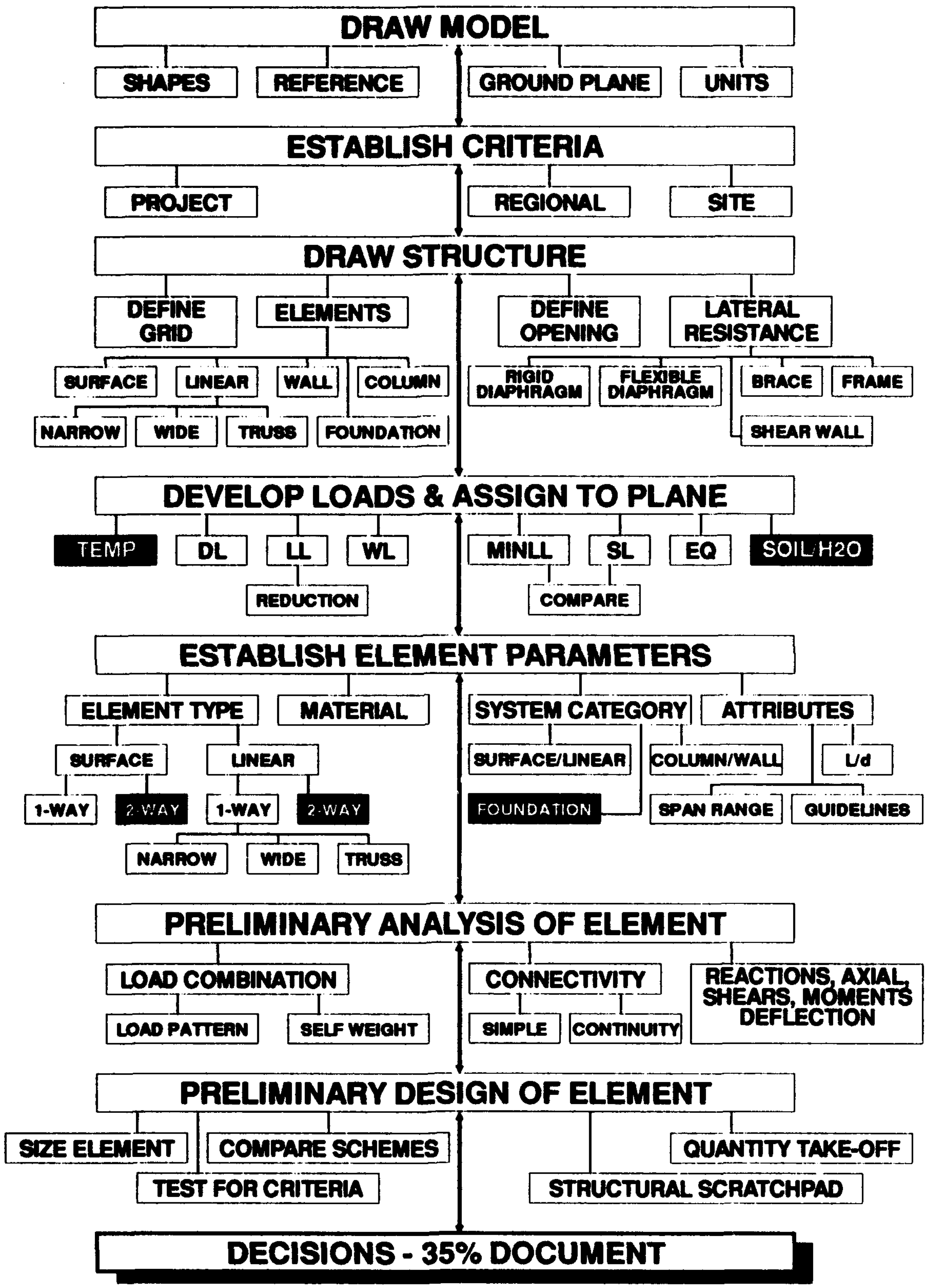
CASM PHILOSOPHY

CASM is a preliminary structural design program that incorporates a Structural Planning philosophy.

Structural Planning is the study of structural system alternatives within the context of each project's unique set of program criteria. The goal of structural planning, and thus CASM, is to select the most appropriate, efficient, and economical structural system which satisfies established program criteria while integrating the mechanical requirements and complementing the intended aesthetics. The structural planning process must begin during the preliminary design phase, when major decisions regarding form, function, and aesthetics are being firmly established. CASM provides fast, interactive "brainstorming," a catalyst for the creative exchange of ideas by the exploration of options that fulfill a desired result. CASM enables the engineer to rapidly answer the question What if?

The structural engineer needs to develop alternatives, approximate proportions, ramifications on the architectural criteria, and implications on cost. Usually, several structural framing schemes are feasible for any given building program. CASM, through the structural planning process, produces an approximate analysis of each solution, to permit the engineer to compare and test each scheme's appropriateness.

CASM is a constantly expanding system that hopes to encompass the myriad of available structural systems, and expound on their attributes as well as their liabilities. CASM is intended to help the engineer in his structural decision-making process. The following flowchart outlines the many facets of CASM and the relationship of its parts.


DRAW MODEL
SHAPES
REFERENCE
GROUND PLANE
UNITS
ESTABLISH CRITERIA
PROJECT
REGIONAL
SITE
DRAW STRUCTURE
DEFINE GRID
ELEMENTS
DEFINE OPENING
LATERAL RESISTANCE
SURFACE
LINEAR
WALL
COLUMN
RIGID DIAPHRAGM
FLEXIBLE DIAPHRAGM
BRACE
FRAME
NARROW
WIDE
TRUSS
FOUNDATION
SHEAR WALL
DEVELOP LOADS & ASSIGN TO PLANE
TEMP
DL
LL
WL
MINLL
SL
EQ
SOIL/H2O
REDUCTION
COMPARE
ESTABLISH ELEMENT PARAMETERS
ELEMENT TYPE
MATERIAL
SYSTEM CATEGORY
ATTRIBUTES
SURFACE
LINEAR
SURFACE/LINEAR
COLUMN/WALL
L/d
1-WAY
2-WAY
1-WAY
2-WAY
FOUNDATION
SPAN RANGE
GUIDELINES
NARROW
WIDE
TRUSS
PRELIMINARY ANALYSIS OF ELEMENT
LOAD COMBINATION
CONNECTIVITY
REACTIONS, AXIAL, SHEARS, MOMENTS DEFLECTION
LOAD PATTERN
SELF WEIGHT
SIMPLE
CONTINUITY
PRELIMINARY DESIGN OF ELEMENT
SIZE ELEMENT
COMPARE SCHEMES
QUANTITY TAKE-OFF
TEST FOR CRITERIA
STRUCTURAL SCRATCHPAD
DECISIONS - 35% DOCUMENT

DESIGN CRITERIA

This is a good time to acquaint you with the CRITERIA pull-down menu and dialog boxes. The Criteria information which you enter will be used for headings and reference on the variety of output files that you will create with CASM for your justification documentation. Information from your design criteria is also used for initial design load values such as wind and snow loads. For each new project, you should start by entering project criteria. This chapter describes the sequence of using the three menu selections on the Criteria pull-down menu--Project, Regional, and Site. For this example you will enter the project criteria data for a new Auditorium at Fort Hauchuca, AZ.

» **Note: For all Criteria dialog window entries you will move the mouse cursor with the mouse and press the left mouse key to select a data box (you may also use the tab key to select data boxes). Once you have selected a box, a flashing vertical cursor will appear. Any information that you type from the keyboard will be inserted at the location of the vertical cursor. You may use the backspace and delete keys to edit your input.**

PROJECT DATA DIALOG WINDOW

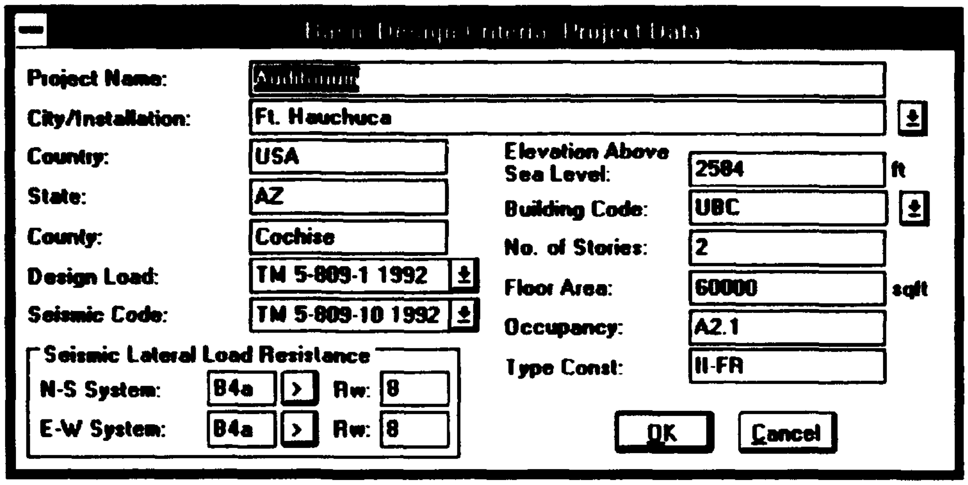
You should always start new projects with the Project dialog window. The Project Name and City/Installation data are used as a heading for all your output files. One feature that you may find useful is the City/Installation database which you can create using the Microsoft Windows Cardfile program. A sample City/Installation cardfile is provided with CASM. To minimize repetitious input, you may create a city/installation database for those cities and/or installations in which the majority of your projects will be located. When you select the City/Installation drop down list, you will have the option of selecting a city or installation from your City/Installation database. All the design data which you have recorded for the selected location will be automatically inserted in your CASM project file. Please refer to the Criteria menu section in the CASM User's Guide for a description of the Cardfile database.

A. Entering Project Criteria data

1. Select CRITERIA from the CASM menu bar, and from the pull-down menu choose PROJECT. The Project Data dialog window will appear.
2. Insert project name: Auditorium

» **NOTE: Avoid pressing the ENTER key after typing the project name. Pressing the ENTER key will automatically close the dialog window and you will have to reopen it. Use the mouse pointer or the TAB key to select input data boxes.**

3. Select the City/Installation data window button. The button to the right of the edit box.
   a. Select Ft. Hauchuca from the pop-up dialog window.



b. Note that stored information from the database is automatically inserted. Verify the inserted information.

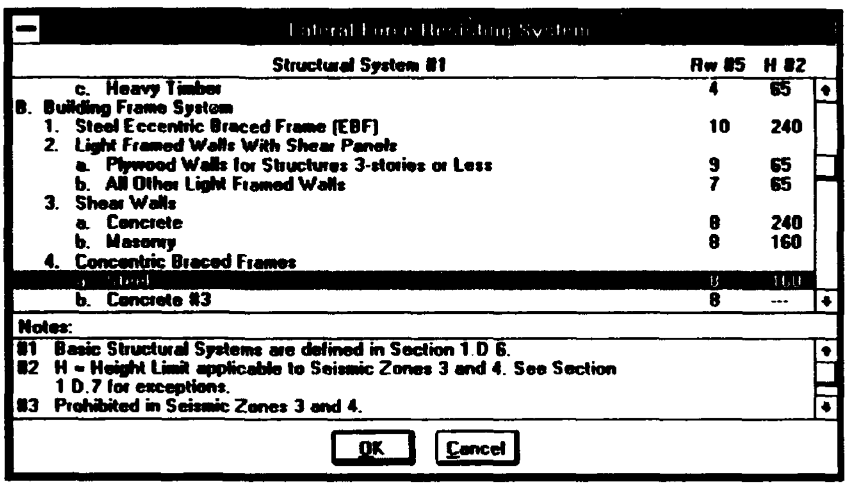
» **Note: If the desired City/Installation name does not appear on the list, you may type in your selected city or installation.**

4. Select the Design Load code used for calculating wind, snow, and minimum roof live loads. You have a choice of the old TM5-809-1 1986 or the new TM5-809-1 1992.

   a. Select TM5-809-1 1986 from the drop down list.

5. Select the Seismic Code data . Only the new TM5-809-10 code is available.

6. Select the Lateral Load Resistance N-S System data window button.



   a. Select B.4.a.: Building Frame System, Concentric Braced Frames, Steel from the pop-up window list.

   b. Click on OK.

7. **Select other data boxes to correct or enter data.**

» **Note: These data items are currently not used by the program for the design and analysis of structural members. However, you may want to insert this information so that it will be included in your hardcopy output.**

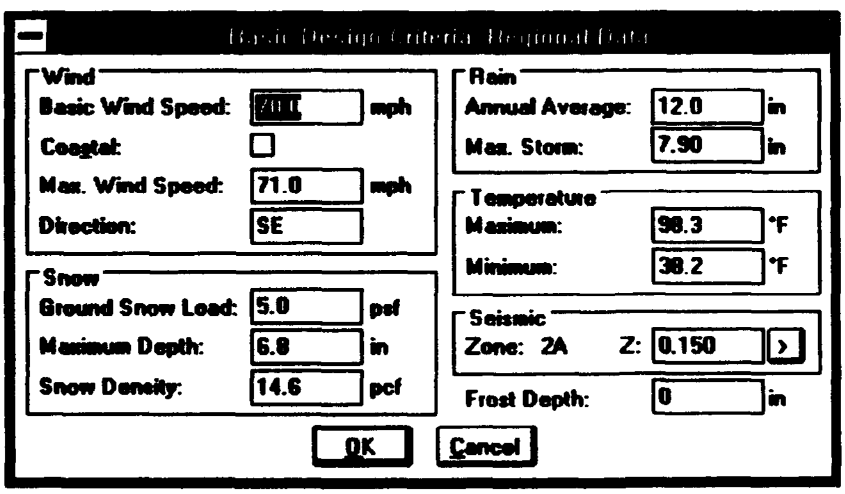
   a. Select the <u>Building Code</u> data box.
      (1) Select UBC from the drop down list.
   b. Select the <u>No. of Stories</u> data box.
      (1) Delete the current value of 1. Type in the number of stories, 2
   c. Select the <u>Floor Area</u> data box.
      (1) Type in the floor area, 60000
   d. Select the <u>Occupancy</u> data box.
      (1) Type in the building occupancy type, A2.1
   e. Select the <u>Type Const</u> data box.
      (1) Type in the type of construction, II-FR

8. Select <u>OK</u> to save your Project Data entries. The Project Data dialog window will disappear.

» **Note: Selecting CANCEL returns you to the main CASM screen without saving changes.**

REGIONAL DATA DIALOG WINDOW

The Regional Data dialog window contains regional meteorological information. Regional information is used for applied loads and design influences on the structural model. Data may be preselected by the Project Data dialog window or overwritten by direct input. The Basic Wind Speed and Ground Snow Load values are the initial values selected for the Wind and Snow Load generation based on the model geometry.



A. Entering Regional Criteria data

1. Select CRITERIA from the CASM menu bar, and from the pull-down menu choose REGIONAL. The Regional Data dialog window will appear.
   a. Note that stored information from the database has been automatically inserted. Verify entries.
2. Select other data boxes to correct or enter data.

» **Note: Data other than Basic Wind Speed, Coastal, Snow Density, and Ground Snow Load, and Seismic Zone are currently not used by the program for the design and analysis of structural members. However, you may want to insert this information so that it will be included in your hardcopy output.**

3. Select OK to save your Regional Data entries. The Regional Data dialog window will disappear.

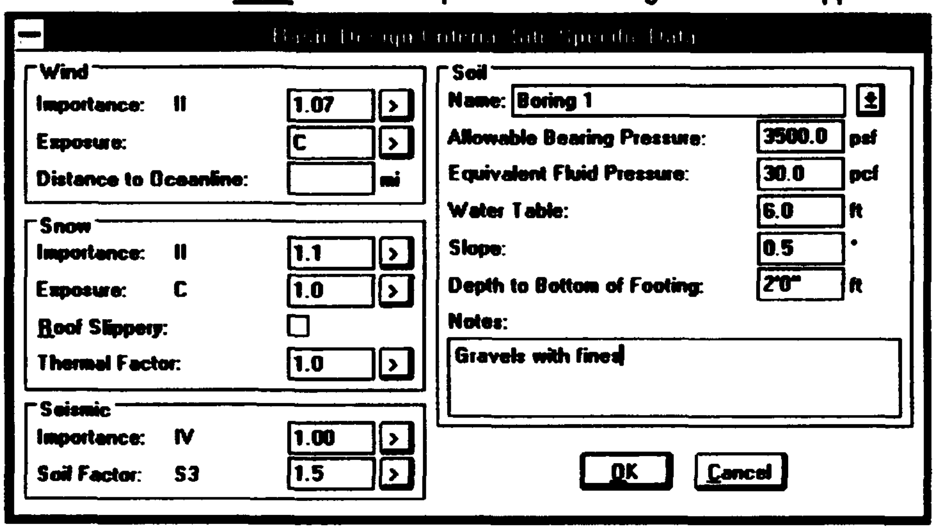
» **Note: Selecting CANCEL returns you to the main CASM screen without saving changes.**

SITE-SPECIFIC DATA DIALOG WINDOW

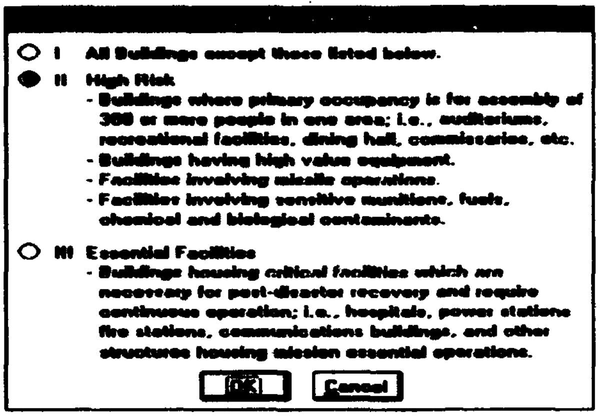
Data here relate to specific design parameters based on building type and location. Only the wind, snow, and seismic data are referenced when you specify a wind, snow, or seismic load on your model. Currently the soil data are not required for structural design, but will be used later for foundation design.

A. Entering Site-Specific Criteria data

1. Select CRITERIA from the CASM menu bar, and from the pull-down menu choose SITE. The Site-Specific Data dialog window will appear.



2. Select the Wind Importance data widow button.
   a. Select the High Risk Importance Factor for the assembly of 300 or more people from the pop-up window.

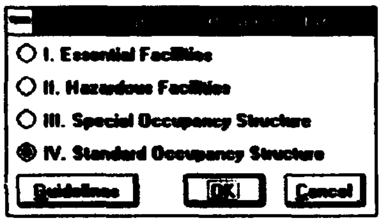


b. Select OK.

c. The Wind Importance Factor changes to 1.07 in the Site-Specific Data dialog window.

d. The Snow Importance Factor changes to 1.1 in the Site-Specific Data dialog window.

3. Select the Seismic Importance data window button.

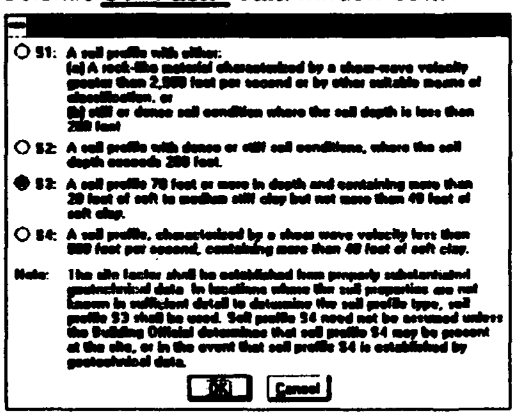


a. Select Importance Factor IV. Standard Occupancy Structure.

b. Select Guidelines for details on the Importance Factor selected.

c. Select OK.

4. Select the Seismic Soil Factor data window button.



a. Select Soil Profile S3.

b. Select OK.

5. Select other data boxes to correct or enter data.

» **Note: The soil data is currently not used by the program for the design and analysis of structural members. However, you may want to insert this information so that it will be included in your hardcopy output.**

» **Note: You may find it easier to: (1) place the mouse pointer in the data box before the current value; (2) press and hold the left mouse key; (3) drag the cursor over the current value to highlight it; and (4) type in the new value. OR (1) double click on a word or number; and (2) type in the new value.**

a. Select the Soil Name data box.

(1) Type in the soil sample name, Boring #1

b. Select the Allow. Bearing Pressure data box.

(1) Type in the allowable bearing pressure, 3500 psf

c. Select the Equiv. Fluid Pressure data box.

(1) Type in the equivalent fluid pressure, 30 pcf

d. Select the Water Table data box.

(1) Type in the water table level, 6 feet

e. Select the Slope data box.

(1) Type in the existing site slope, 0.5°

f. Select the Depth to Bottom of Footing data box

(1) Type in a depth of 2.0 feet.

g. Select the Notes data box.

(1) Type in the type of soils present at the site: Gravels with fines.

» **Soil data is automatically saved for each unique soil name. Use the Soil Name drop down list to view and edit other soil data.**

6. Select OK to save your Site-Specific Data entries. The Site-Specific Data dialog window will disappear.

» **Note: Selecting CANCEL returns you to the main CASM screen without saving changes.**

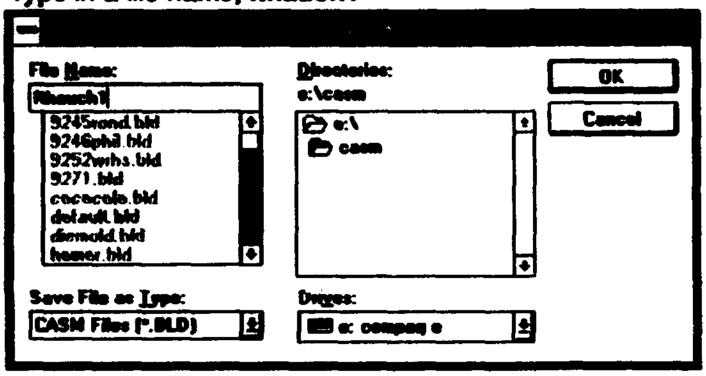
SAVING PROJECT DATA

Up to now you have been saving your criteria entries in CASM; however, you will need to save all your project data in a project file on the hard disk or on a floppy disk. You should get into the habit of saving your work in the project file on the hard disk frequently as you input data. For example, you have spent several hours working on a project in CASM without saving data. Suddenly there is a momentary loss of electric power. All of your work will be lost and you will have to repeat all of your inputs. If you save your work frequently, a momentary power loss will not be catastrophic.

A. To save project data:

1. Select FILE from the CASM menu bar, and from the pull-down menu choose SAVE. If the project file is 'untitled,' the Save As File Name dialog window will appear, otherwise the saved project file will be updated.

   a. Type in a file name, fthauch1



» Note: The name is limited to eight characters or less. A .bld extension is automatically added to the name you choose. Later, when running CASM and you look for your project file, it will be listed as FTHAUCH1.BLD.

   b. Select OK.

   c. An hourglass symbol will appear as your project file is saved.

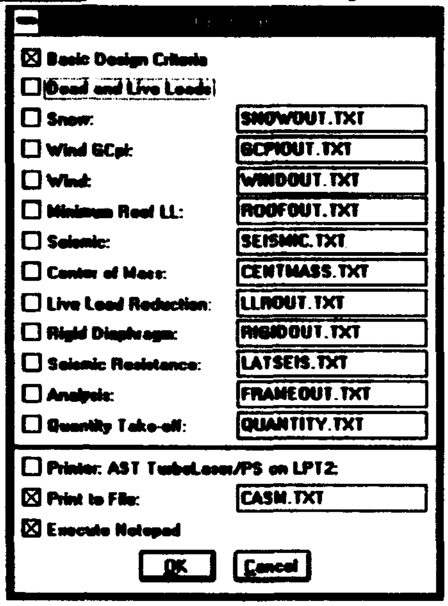
» Note: CASM automatically checks for other project files with the same name before it saves your project file. If there is another project file with the same name, you will be reminded so that you will not overwrite that project file.

PRINTING PROJECT CRITERIA DATA

At any time you may print out a copy of your design criteria data. You may also print the design criteria to a file where you can edit the criteria before printing it or transfer the file to another computer for printing. For training purposes we will assume that you do not have a printer connected to your computer, so we will describe how to print the criteria data to a file.

A. To print project data to a file:

1. **Select FILE from the CASM menu bar, and from the pull-down menu choose PRINT DATA. The Print Data dialog window will appear.**



   a. Initially the Basic Design Criteria selection box is selected (an X is in the selection box).

   b. Deselect the Dead & Live Loads selection by placing the mouse pointer on the selection box and pressing the left mouse key. The X will disappear.

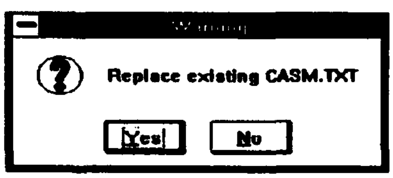
   c. Make sure all other data files are not selected.

   d. The Print to File selection box is already selected, and a default file name is in the File name box, CASM.TXT.

» **Note: Because the output file can be easily created by CASM, we recommend that you use the default file name and overwrite existing output files rather than generate new output files every time you desire to print to a file.**

   e. Select Execute Notepad to run notepad after writing the output file.

   f. Select OK.



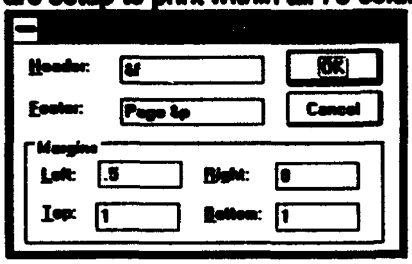
   g. Select YES to overwrite the existing CASM.TXT file.

2. You will automatically be placed in the NOTEPAD Application program window where you may edit the CRITERIA data before you print it. The NOTEPAD Application program window is shown below.

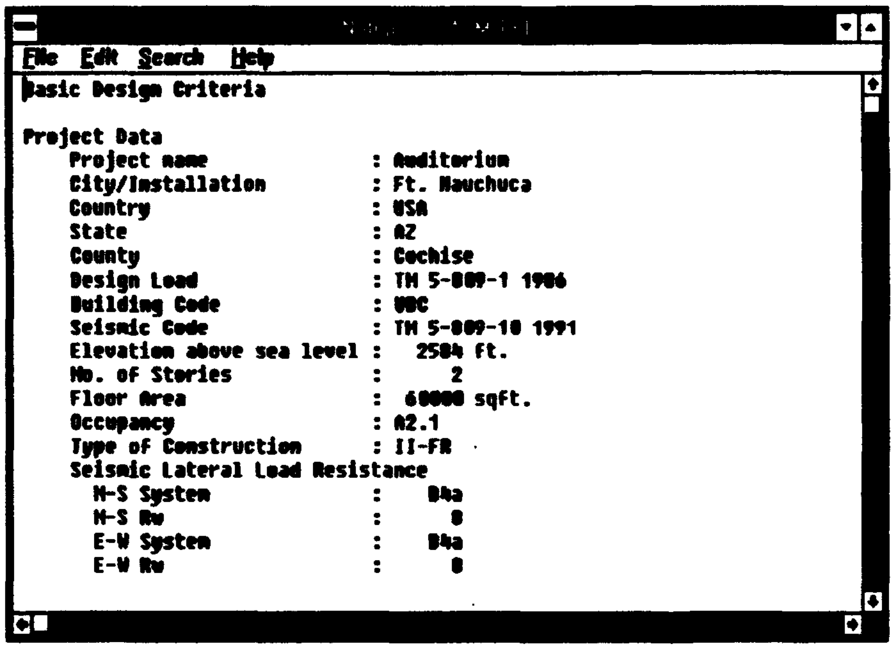
a. Activate the side bar scroll bar to move up/down the entire page or use the [Page Down]/[Page Up] keys on the keyboard.

b. To edit text, place the mouse pointer at the location where you want to modify the text and press the left mouse key. A flashing vertical cursor will appear at that point, and you may type in your changes. Use the [Backspace] and [Delete] keys to eliminate characters.

c. To print text, first select Page Setup from the File pull-down menu. Change the left margin to 0.5 inches and right margin to 0 since CASM output files are setup to print within all 75 columns of a printer.



d. To print text, select FILE from the NOTEPAD menu bar, and from the pull-down menu choose PRINT. The file will be printed.



An example of the output format is as follows:

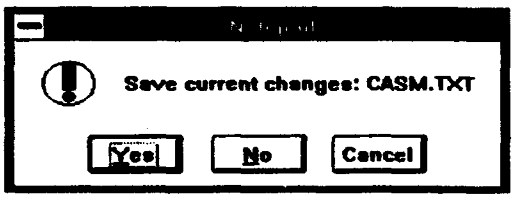
```
Basic Design Criteria

Project Data
    Project name                : Auditorium
    City/Installation           : Ft. Hauchuca
    Country                     : USA
    State                       : AZ
    County                      : Cochise
    Design Load                 : TM 5-809-1 1986
    Building Code               : UBC
    Seismic Code                : TM 5-809-10 1991
    Elevation above sea level   :   2584 ft.
```

```
    No. of Stories              :      2
    Floor Area                  :  60000 sqft.
    Occupancy                   : A2.1
    Type of Construction        : II-FR
    Seismic Lateral Load Resistance
      N-S System                :    B4a
      N-S Rw                    :      8
      E-W System                :    B4a
      E-W Rw                    :      8

Regional Data
  Wind
    Basic Wind Speed            :   70.0 mph
    Coastal                     :     No
    Maximum Wind Speed          :   71.0 mph
    Wind Direction              :     SE
  Snow
    Ground Snow Load            :    5.0 psf
    Maximum Snow Depth          :    6.8 in.
    Snow Density                :   10.0 pcf
  Rain
    Average Annual Rainfall     :   12.0 in.
    Maximum Rainfall            :    7.9 in.
  Temperature
    Maximum Temperature         :   98.3 F
    Minimum Temperature         :   38.2 F
  Seismic Zone   : 2A           :  0.150
  Frost Depth                   :      0 in.

Site Specific Data
  Wind
    Exposure                    :      C
    Importance   : II           :   1.07
  Snow
    Exposure     : C            :   1.00
    Importance   : II           :   1.10
    Roof Smooth                 :     No
    Thermal Factor              :    1.0
  Seismic
    Importance   : IV           :   1.00
    Soil Factor : S3            :    1.5
  Soil Name : Boring #1
    Allowable Bearing Pressure: 3500.0 psf
    Equivalent Fluid Pressure :   30.0 pcf
    Water Table                 :    6.0 ft.
    Slope                       :    0.5
    Depth to Bottom of Footing:      2.0 ft.
    Gravels with fines

Notes
  Importance Factor for Snow and Wind:
    II    High Risk
        - Buildings where primary occupancy is for assembly of
          300 or more people in one area; i.e., auditoriums,
          recreational facilities, dining hall, commissaries, etc.
        - Buildings having high value equipment.
        - Facilities involving missile operations.
        - Facilities involving sensitive munitions, fuels,
          chemical and biological contaminants.
  Wind Exposure Category:
    Exposure C:
      Open terrain with scattered obstructions having heights
      generally less than 30 ft.
  Snow Exposure Category:
    Exposure C:
      Snow removal by wind cannot be relied on to reduce roof
      loads because of terrain, higher structures, or several
      trees nearby.
  Snow Thermal Factor:
    Heated structure.
    * These conditions should be representative of those that are likely
    to exist during the life of the structure.
  Seismic Lateral Load Resistance System:
    B. Building Frame System
        4. Concentric Braced Frames
            a. Steel
                Height limit #2: 160
    #1 Basic Structural Systems are defined in Section 1.D.6.
    #2 H = Height Limit applicable to Seismic Zones 3 and 4. See Section
       1.D.7 for exceptions.
    #5 See Section 1.E.3 for combination of Structural System.
  Importance Factor for Seismic:
    I. Essential Facilities
      Hospitals and other medical facilities having surgery and emergency
      treatment areas.
      Fire and police stations.
      Tanks or other structures containing, housing or supporting water
      or other fire-suppression materials or equipment required for the
      protection of essential or hazardous facilities, or special
      occupancy structures.
      Emergency vehicle shelters and garages.
```

```
      Structures and equipment in emergency preparedness centers.
      Stand-by power generating equipment for essential facilities.
      Structures and equipment in communication centers and other
      facilities required for emergency response.
    II. Hazardous Facilities
      Structures housing, supporting or containing sufficient quantities
      of toxic or explosive substances to be dangerous to the safety of
      the general public if released.
    III. Special Occupancy Structure
      Covered structures whose primary occupancy is public assembly -
      capacity more than 300 persons.
      Buildings for schools (through secondary) or day-care centers -
      capacity more than 250 students.
      Buildings for colleges or adult education schools - capacity more
      than 500 students.
      Medical facilities with 50 or more resident incapacitated patients,
      but not included above.
      Jails and detention facilities.
      All structures with occupancy more than 5000 persons.
      Structures and equipment in power generating stations and other
      public utility facilities not included above, and required for
    IV. Standard Occupancy Structure
      All Structures having occupancies or functions not listed above.
  Seismic Soil Factor:
    S3: A soil profile 70 feet or more in depth and containing more than
        20 feet of soft to medium stiff clay but not more than 40 feet of
        soft clay.
    The site factor shall be established from properly substantiated
    geotechnical data. In locations where the soil properties are not
    known in sufficient detail to determine the soil profile type, soil
    profile S3 shall be used. Soil profile S4 need not be assumed unless
    the Building Official determines that soil profile S4 may be present
    at the site, or in the event that soil profile S4 is established by
    geotechnical data.
```

End of example output format.

1. **Return to the CASM program window by moving the mouse pointer to the CONTROL menu box in the top left corner of the screen. Double click the left mouse key.**
   a. **If you have made any changes to the text file, you will be prompted to save them. For this example, select NO.**



   b. **This returns you to the CASM program window screen.**

You are now ready to begin your structural model and create specific load cases based upon the three CRITERIA data sets.

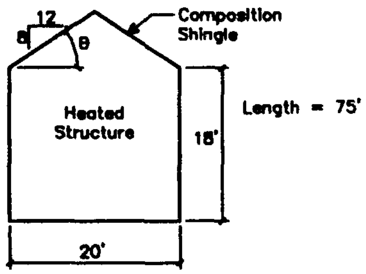
LOADS

SNOW LOADS

This section includes examples of snow load design for (1) a gable roof, (2) an arched roof with several parameter variations, (3) a lean-to roof adjacent to a taller roof with drifted and sliding snow considerations, and (4) a multiple-gabled roof.

EXAMPLE ONE: GABLE ROOF

Given: This gable roof example is taken from page E-1 of TM 5-809-1 1986. It is a dormitory building sited among several nearby pine trees. It is a heated structure with composition shingles located at Westover AFB, MA. Dimensional data are given, and there are no adjacent structures.



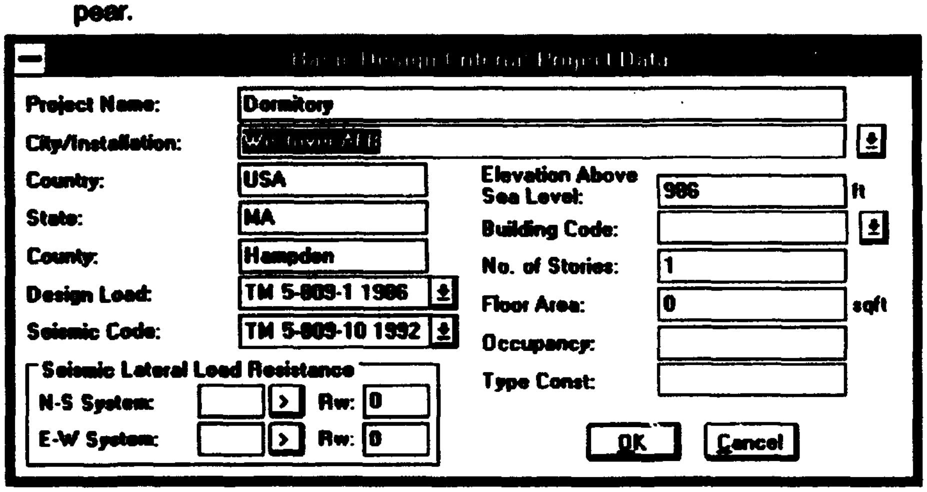
Required: Determine the balanced and unbalanced roof snow loads.

Solution:

A. Establish Criteria

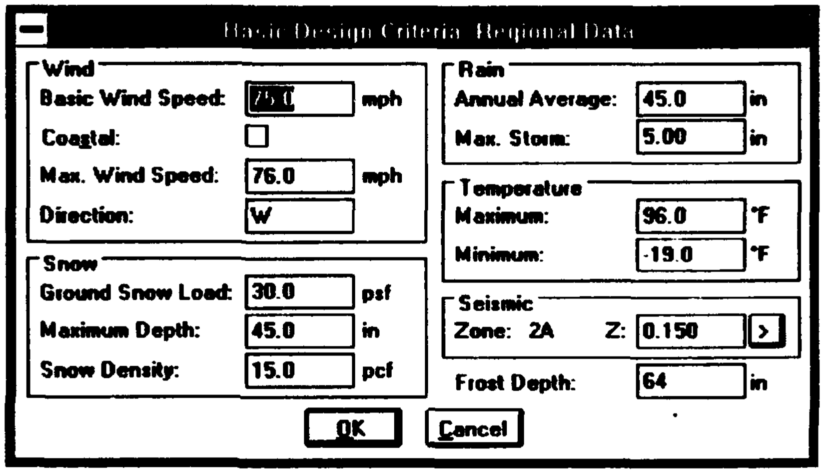
» **Select NEW from the FILE pull-down menu to start a new project file.**

1. Select CRITERIA from the menu bar, and from the pull-down menu choose <u>PROJECT</u>. The PROJECT Criteria pop-up dialog window will appear.

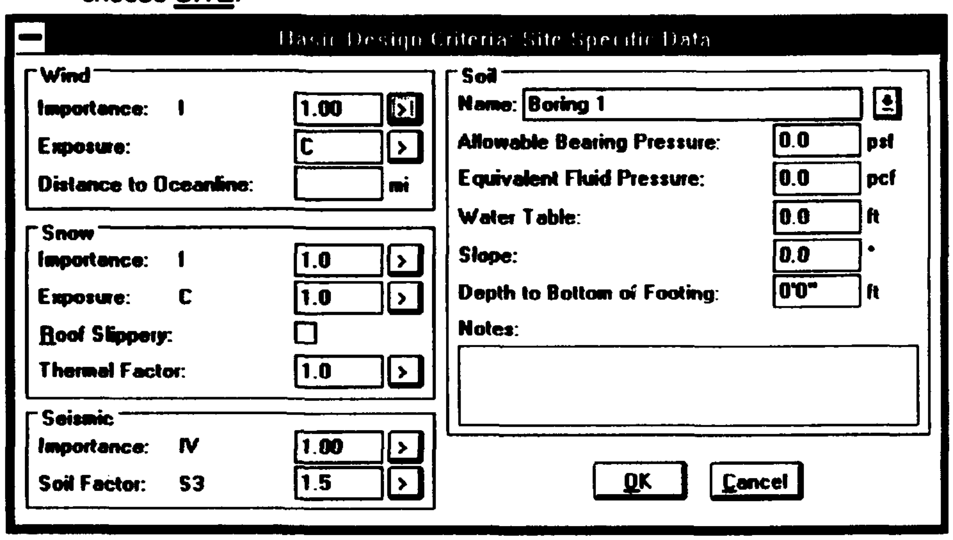


2. Insert project name: Dormitory.
3. Move mouse arrow to the CITY/INSTALLATION drop down list button. Select Westover AFB from the drop down list. Note that stored information from the database is automatically inserted.
4. Select TM5-809-1 1986 from the Design Load drop down list.
5. No other user-inputted data is required here, so select OK and return to the basic WINDOW screen.

6. Select CRITERIA from the menu bar, and from the pull-down menu choose REGIONAL. Note that the GROUND SNOW LOAD has already been inserted, since the city/installation came from the database. No other data is required to solve this problem, so select OK and return to the basic WINDOW screen.

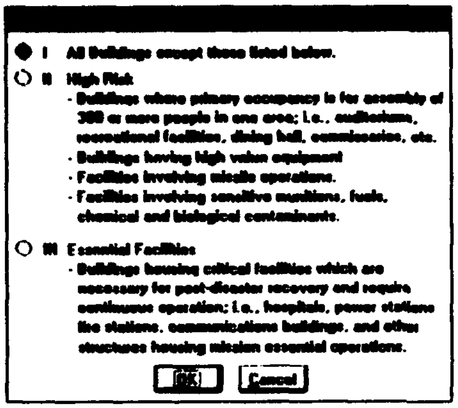


7. Select CRITERIA from the menu bar, and from the pull-down menu choose SITE.

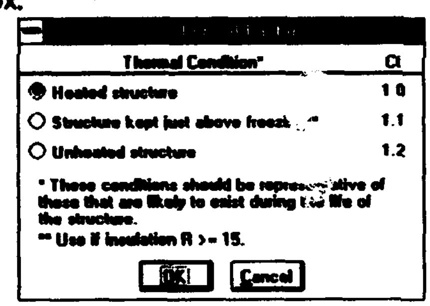


8. Move the mouse arrow to the box for SNOW IMPORTANCE factor. Click the left mouse key to activate the pop-up dialog window and make an appropriate selection of a factor. When the desired circle for factor I is high-

lighted, move the mouse pointer to OK and click the left mouse key. The chosen factor will appear in the data box.



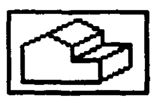
9. Repeat the previous step for EXPOSURE and select an exposure category from the pop-up dialog window. Highlight category C and click on OK. A '1.0' and a 'C' are automatically placed in the proper data box.
10. A composition roof is not considered a slippery surface, so leave the box unchecked.
11. Move mouse pointer to the data window button for THERMAL FACTOR and click on the left mouse key. Highlight HEATED STRUCTURE in the pop-up dialog window and click on OK. A thermal factor of 1.0 is placed in the data box.



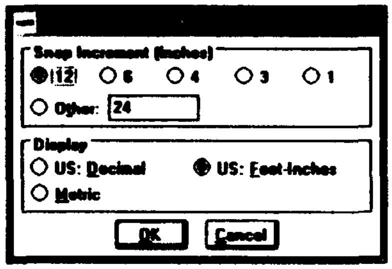
12. This completes the required input of data on the SITE SPECIFIC dialog window. Select OK and return to the CASM program window. You have now completed entry into CRITERIA.

B. Draw volumetric model

1. Select the Draw Model button to display the Draw Model tool palette, if it is not already displayed.



2. Establish general layout requirements.
   a. Select the DEFINE UNITS command from the Layout pull-down.



(1) Set the <u>SNAP INCREMENT</u> to 12 inches.

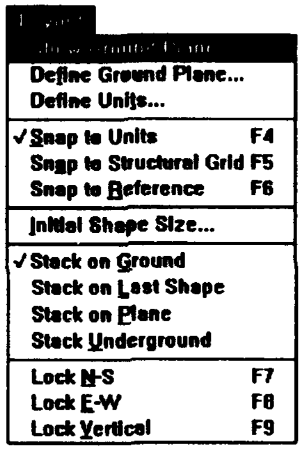
(2) Set the Display to Feet-Inches.

(3) Click on <u>OK</u>.

b. Turn on <u>SNAP TO UNITS</u> from the Layout pull-down.

» **Note: Snap To Units is on when there is a checkmark next to the command or the icon is highlighted.**

c. Turn on <u>SHOW GROUND PLANE</u> from the Layout pull-down.

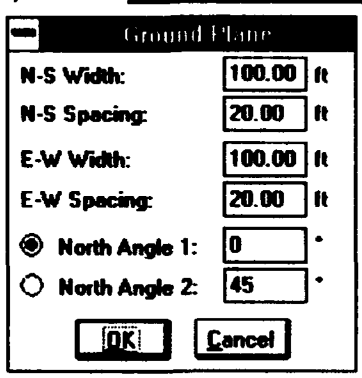


d. Select <u>DEFINE GROUND PLANE</u> command from the Layout pull-down.

» **Note: Make sure the ground plane dimensions are larger than the overall building dimensions.**

(1) Set the <u>NS and EW WIDTH</u> to 100 ft.

(2) Set the <u>NS and EW SPACING</u> to 20 ft.

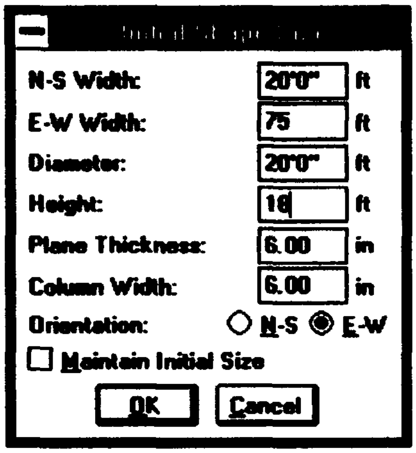


» **Hint: Use a larger spacing when using the single screen windows graphics library to decrease the number of lines drawn.**

(3) Set the NORTH ANGLE 1 to 0 degrees. (See reference manual)

(4) Click on OK.

e. Select INITIAL SHAPE SIZE command from the Layout pull-down.



(1) Set the NS WIDTH to 20 ft.

(2) Set the EW WIDTH to 75 ft.

(3) Set the HEIGHT to 18 ft.

(4) Set the ORIENTATION to E-W since the ridge runs parallel to the east/west dimension.

(5) Click on OK.

f. Turn on STACK ON GROUND from the Layout pull-down since we will want the first object to sit on the ground.

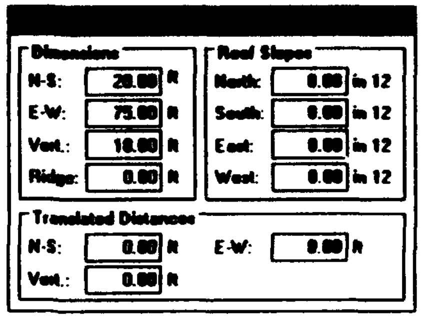
» **Note: The current stack mode's command has a checkmark next to it and the icon is highlighted.**

g. Make sure no directions are locked. This allows shapes, edges, and vertices to be moved in all three orthogonal directions.

» **Note: A direction is locked if there is a checkmark next to the command or the icon is highlighted.**

3. Create the first floor building volume.

a. Select the CUBE icon from the Draw Model tool palette or from the Shapes pull-down. The shape will appear on ground plane to the proportions selected under Initial Shape Size. The Dimensions pop-up dialog window will appear with all the dimensions of the shape indicated.



(1) Drag the shape to a location on the ground plane by moving the mouse.

» **Note: Moving the mouse right/left moves the object east/west, while moving the mouse away/toward moves the object north/south.**

(2) Click the left mouse key to fix the location of the cube. A duplicate shape appears and is movable.

(3) Double click the right mouse key to exit the command and stop adding shapes to the ground plane.

» **Note: Double clicking the right mouse key in any graphic command will exit the command.**

4. Create the gable roof form.

a. Turn on STACK ON LAST SHAPE from the Layout pull-down. The next *selected shape will sit on top of the last shape.*

b. Select the PRISM icon from the Draw Model tool palette or from the Shapes pull-down. The prism will appear on the last shape drawn, and a pop-up dimensions dialog window will also appear. Click left mouse key to insert the prism.

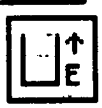
» **Note: You do not need to double click the right mouse key to exit the command since you cannot stack another shape on top of a prism.**

c. Select the DEFINE UNITS command from the Layout pull-down.

(1) Set the SNAP INCREMENT to 4 *inches to make it easier to set the desired roof slope.*

(2) Click on OK.

d. LOCK the NS and EW directions from the Layout pull-down. We only *want the vertical movement of the ridge allowed.*



e. Select the DRAG EDGE command from the Edit pull-down. Solid square handles appear on each edge of the constructed model.

» **Note: Only visible plane's edges can be dragged, and edges can only be dragged in directions where all planes connecting to the edge remain planar.**

(1) Select the ridge edge handle by clicking the left mouse key when the pointer is over the handle.

(2) Hold the right mouse key down while dragging the mouse up and down until the roof slope appears as 8.00 in 12 in the dimensions dialog box. The vertical height of the prism becomes 6'-8".

>> Note: Holding the right mouse key down while dragging is the vertical direction for all drag commands.

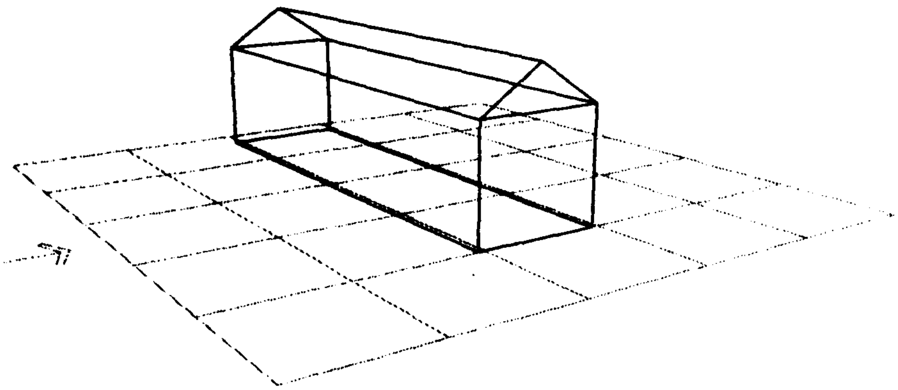
>> Note: If you are having trouble dragging the edge to the 8 in 12 slope with the mouse, use the keyboard instead. While holding down the [Alt] key, press the up and down arrow keys to drag vertically. Press the [Enter] key to fix the location. Press the [Esc] key to exit the command.

(3) Click the left mouse key to fix the position of the dragged edge.

(4) Double click the right mouse key to exit the command. Otherwise, you could now drag another edge.

f. UNLOCK both the NS and EW directions.

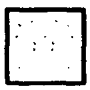
5. This completes the model for this example.

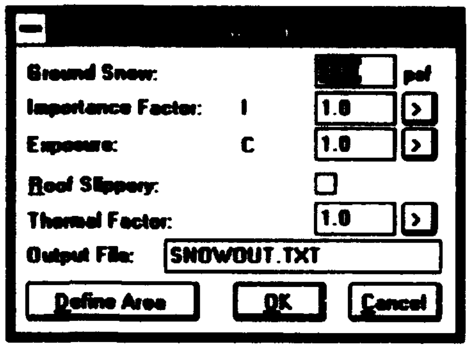


C. Develop snow loads on the roof.

1. Select the LOADS AND DESIGN button to display the Loads and Design tool palette.

2. Select SNOW LOADS from the Loads pull-down menu or from the snow icon within the Loads Tool Palette. A snow loads pop-up dialog window will appear.





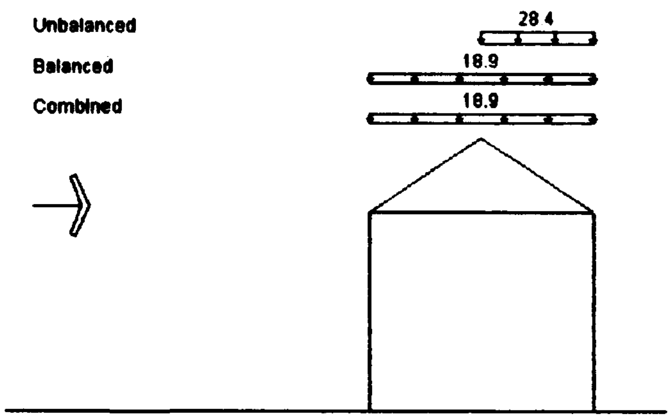
3. The Snow Loads dialog window contains the decisions from completion of the Criteria windows that were previously entered for determination of snow design loads.
   a. Change any of the parameters upon which snow calculations will be based. Any value within the window can be revised or added at this time.

» Note: Pop-up windows will appear to help make decisions regarding Importance & Exposure factors, if these data window buttons are selected for modification.

   b. If satisfied, click on OK and the roof snow load calculations will automatically begin.
   c. A warning box may appear to prompt you if you will replace an existing output file.

» Note: A pop-up dialog window will keep you informed of the program's progress to assure you that it is still calculating and has not stopped processing.

4. The building plan and a section elevation will appear upon completion of snow loads calculations. The various snow loads calculated will appear on the screen above the roof with magnitudes and descriptors.



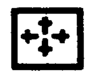
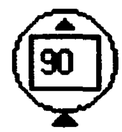
Snow Loads (psf)

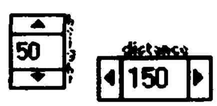
   a. Drag the mouse to move the horizontal line in plan to where you want the section cut made.
   b. Click on the left mouse key to redraw the section at that location.

   c. Double click the right mouse key to fix the section cut location selected.

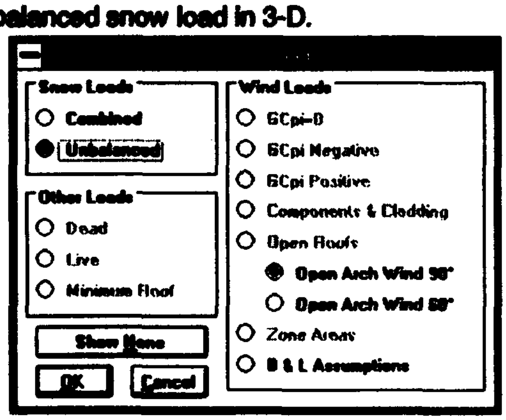
D. Manipulation of building model and its snow loads.

1. Zoom the graphics on the screen.
   a. Move the mouse pointer to the left arrow of the Distance tool.
   

   b. Click on the left mouse key to decrement the viewing distance toward you.
   c. Press and hold the left mouse key while dragging the mouse left and right to zoom in and out.
   d. Release the mouse key when the desired zoom is achieved.
   e. Use the right arrow of the Distance tool to zoom out.
2. Zoom a window on the screen.
   a. Select the Zoom Window icon button.
   
   b. Select one corner of the window with the left mouse button.
   c. Select the opposite corner of the window with the left mouse button.
3. Pan the screen image.
   a. Select the Pan icon button with the mouse pointer.
   
   b. Drag the mouse to pan the view.
   c. Click the left mouse key to save the Pan position.
4. Display a previous view by selecting the previous view icon button.
   
5. View the model and its balanced snow load in wireframe 3-D.
   
   a. Select the View pulldown menu and select PERSPECTIVE (3D). Transparent and solids are also possible.
6. Rotate the 3-D model view and its snow load.
   

   a. Click the left mouse key at the location on the circle where you want to view from.
   b. To dynamically rotate the 3D view, hold the left mouse key down while moving the mouse pointer, and drag the black arrow around the circle in the Viewpoint window.
   c. Release the left mouse key when the desired viewing angle is achieved.
   

   d. You can also change the viewing height and distance similar to step 1.
   e. You may also type in the desired angle, height, and distance.
7. View the unbalanced snow load in 3-D.



a. Select SHOW LOADS from the View pull-down menu and a dialog window of choices will appear.

b. Click the left mouse key on the UNBALANCED SNOW LOAD.

c. Select OK. The 3-D snow pattern will be redrawn to reflect your choice.

» **Note: The wind direction for this unbalanced case is also shown with the model and will be in the direction of the last section cut.**

E. Generation of hard copies.

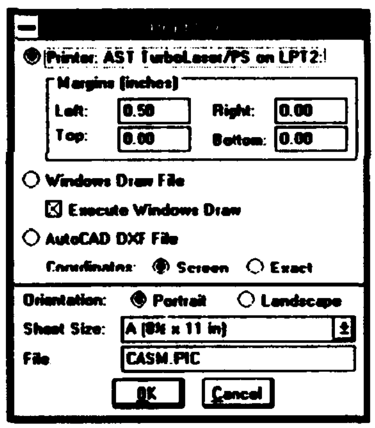
1. Return to the 2-D section cut.

a. Select SECTION from the View menu.

» **Note: The 2-D section that appears is dependent on the viewing direction of the 3-D model. To obtain a section cut perpendicular to the ridge, rotate the 3-D view so that you are looking at a view that is approximately the desired section cut. It is not necessary to adjust the height or distance.**

2. Print the screen image.

a. Select PRINT SCREEN from the File pull-down menu on the CASM menu bar. A pop-up dialog window will appear.



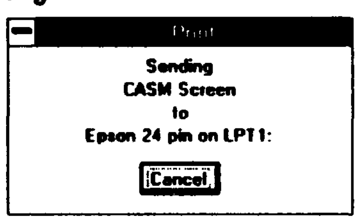
b. Select PRINTER to print directly to the printer.

» **Note: Make sure your printer is on-line and ready to print before selecting OK.**

c. Set the margins.

d. Select the desired page orientation.

e. Select OK to begin printing. A dialog window will appear to allow you to cancel the printing.

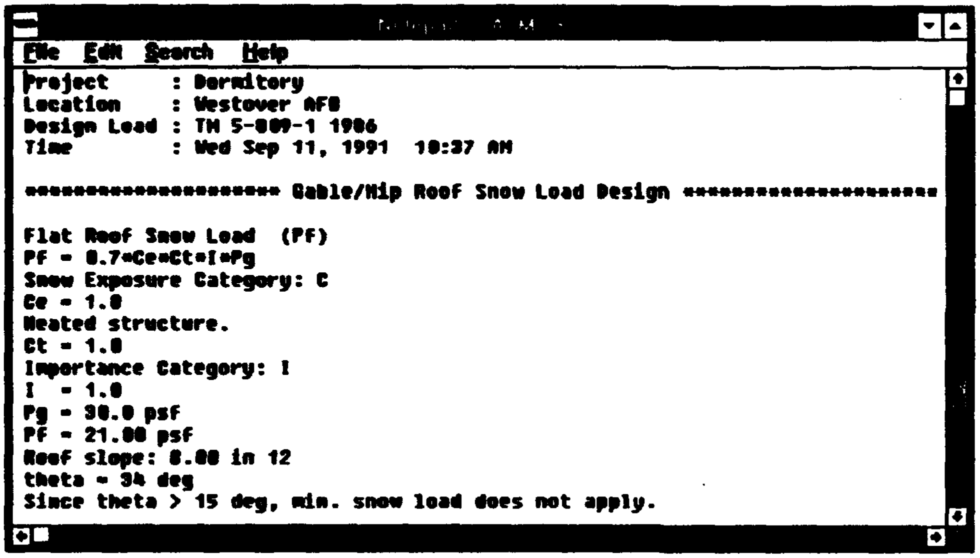


» **If you get an error message while printing, refer to the Reference Manual chapter 6 for information on resuming the print.**

3. **Review the snow load calculations on the screen.**
   a. **Select PRINT DATA from the File pull-down menu. The Print Data dialog window will appear.**
   b. **Select SNOW to include the snow load text output file.**
   c. **Deselect all other print data output options.**
   d. **Select PRINT TO FILE and enter an appropriate file name.**
   e. **Select EXECUTE NOTEPAD to run Notepad.**

» **Note: We could have selected Print to Printer to obtain a hard copy of the output.**

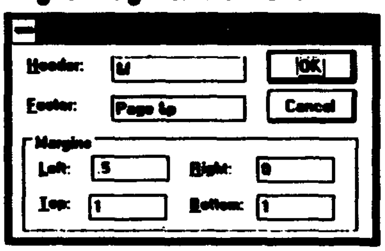
   f. **Select OK to generate the file and execute Notepad. Notepad will appear on the screen.**



File Edit Search Help

```
Project      : Dormitory
Location     : Westover AFB
Design Load  : TM 5-809-1 1986
Time         : Wed Sep 11, 1991  10:37 AM

******************** Gable/Hip Roof Snow Load Design ********************

Flat Roof Snow Load  (PF)
PF = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: C
Ce = 1.0
Heated structure.
Ct = 1.0
Importance Category: I
I  = 1.0
Pg = 30.0 psf
PF = 21.00 psf
Roof slope: 8.00 in 12
theta = 34 deg
Since theta > 15 deg, min. snow load does not apply.
```

4. **Print the snow load calculations.**
   a. **Select PAGE SETUP from the File pull-down menu. Set the left margin to 0.5 and the right margin to 0.0 inches.**



   b. **Select PRINT from the File pull-down menu to generate a hard copy.**

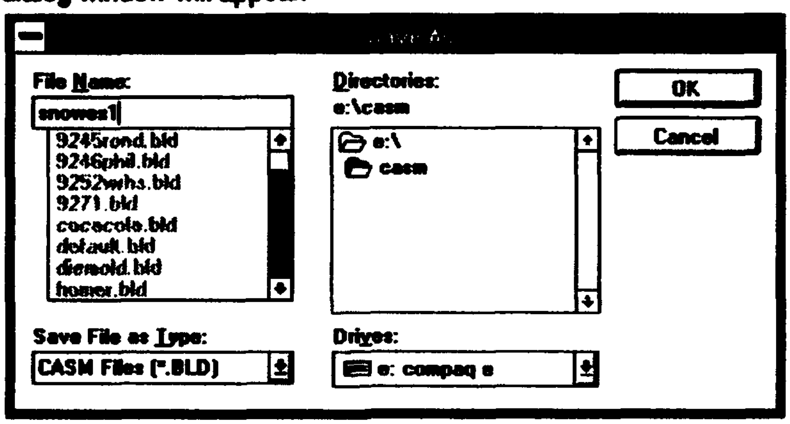
5. **Select EXIT from the File pull-down menu to close Notepad. Redraw the screen by clicking on Distance, if using the single screen version.**

Example 1 sample output (SNOWEX1.TXT):

```
Project     : Dormitory
Location    : Westover AFB
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991  12:32 PM

********************* Gable/Hip Roof Snow Load Design *********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: C
Ce = 1.0
Heated Structure.
Ct = 1.0
Importance Category: I
I  = 1.0
Pg = 30.0 psf
Pf = 21.00 psf
Roof Slope: 8.00 in 12
Theta = 34 deg
Since theta > 15 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+------------------------+
|     Pf = 21.00 psf     |
+------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 0.90
+------------------------+
|     Ps = 18.90 psf     |
+------------------------+

Unbalanced Snow Load (Punbal)
Since 15 deg < theta < 70 deg, unbalanced condition applies.
Punbal = 1.5*Ps/Ce
+------------------------+
|  Punbal = 28.35 psf    |
+------------------------+
```

F. Save the building model with its snow loads applied for future reference.

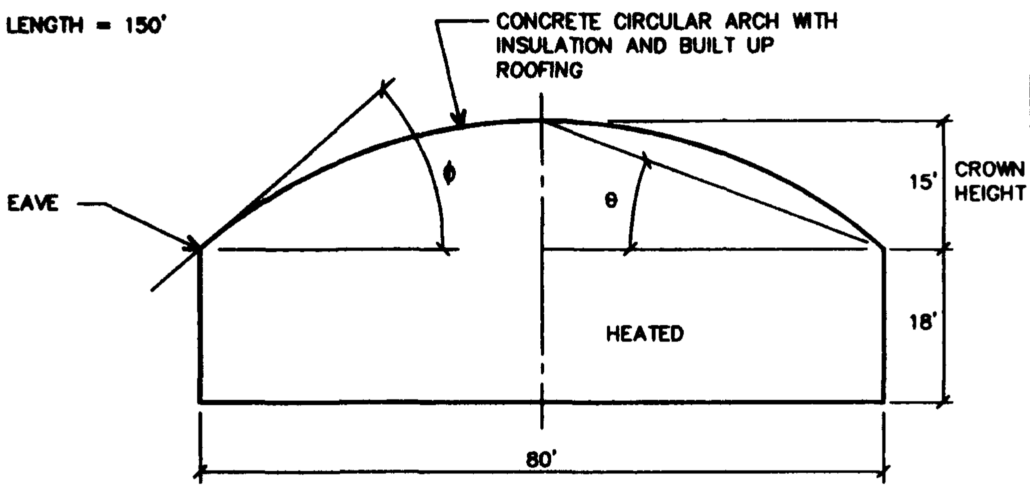
1. **Select SAVE from the File pull-down on the CASM menu bar. A pop-up dialog window will appear.**



2. **Type in the desired Filename.**
3. **Click the left mouse key on OK.**
4. **The saved file can now be accessed as needed from OPEN in the File pull-down.**

» **Note: The extension .bld is automatically added to the filename.**

Given: This arched roof example is taken from page E-4 of TM 5-809-1 1986. It is a theater (greater than 300 occupancy) sited in a windy area with a few nearby coniferous trees. It is the tallest structure in a recreational complex. The building is heated and the roof is sheathed with built-up roofing. It is located in Milwaukee, WI (not Chicago as stated in the TM).



Required: Determine the balanced and unbalanced snow loads.

Solution: An abbreviated discussion is given here since the steps basically repeat those of example one.

A. Establish Criteria.

1. Input the following data into the PROJECT, REGIONAL, and SITE CRITERIA dialog windows:

PROJECT	Project name	: Theater
	City/Installation	: Milwaukee
	State	: WI
	Design Load	: TM 5-809-1 1986
REGIONAL	Ground Snow Load	: 35 psf
SITE	Snow Importance	: category II
	Snow Exposure	: category B
	Thermal Factor	: Heated
	Roof Slippery	: no

B. Draw volumetric model

» **Note: There are many ways to construct the model. This example will illustrate a different approach to some of the steps to emphasize the variety of options.**

1. Select DRAW MODEL from the menu bar.
2. Establish general layout requirements which are different than previously established.
   a. Use the following:

DEFINE UNITS (snap increment)			: 12 inches
SNAP TO UNITS			: on
SHOW GROUND PLANE			: on
DEFINE GROUND PLANE			
	WIDTH	N-S	: 100 feet
		E-W	: 200 feet
	SPACING	N-S	: 20 feet
		E-W	: 20 feet
INITIAL SHAPE SIZE			
	N-S WIDTH		: 20 feet
	E-W WIDTH		: 20 feet
	HEIGHT		: 20 feet
	ORIENTATION		: E-W
STACK ON GROUND PLANE			: on
DIRECTIONS LOCKED			: none

» **Note: The ground plane grid is now rectangular for this example. You may wish to increase the viewing distance to make the entire ground plane visible.**

3. Create the first floor building volume.
   a. Select CUBE and fix the initial shape location in the northwest corner of the ground plane.
   b. Modify the initial object dimensions to the required building proportions.
      (1) Select DRAG PLANE from the Edit pull-down. Solid square handles will appear at the centroid of each visible plane on the object.

» **Note: Only visible planes can be dragged. Planes can only be dragged in a direction perpendicular to the plane. Also, only a plane that has all adjacent planes perpendicular to it can be dragged.**

      (2) Select the top plane handle with the left mouse key when the cursor is over the handle. The selected plane will be highlighted and the pop-up Dimensions dialog window will appear with the object's dimensions inserted.
      (3) Drag the mouse toward and away to vertically change the top plane's height above the ground plane to 18 feet.

» **Note: Always move the mouse toward and away regardless of the plane's required direction of movement. There is no need to hold down the right mouse key to drag a plane vertically.**

(4) Click the left mouse key to fix the location of the plane. The handles will reappear to allow for additional drag plane operations.

(5) Select the south plane handle with the mouse as in step 2.

(6) Drag the mouse toward and away until the N-S dimension shows 80 feet in the dialog window.

(7) Click the left mouse key to fix the location of the south plane.

(8) Select the east plane handle with the mouse.

(9) Drag the mouse toward and away until the E-W dimension shows 150 feet.

(10) Click the left mouse key to fix the location of the east plane.

(11) Double click the right mouse key to exit the Drag Plane command. This completes the first floor volume.

4. Create the barrel vault roof form.

a. Turn on STACK ON PLANE from the Layout pull-down menu to select the appropriate plane to receive the barrel vault.

» **Note: STACK ON LAST SHAPE could also have been used for this example since the barrel vault connects directly to the top plane of the first floor volume.**

b. Set the INITIAL SHAPE SIZE to 15 feet in height to reflect the required crown height.

» **Note: Barrel vault crown height cannot be modified from a Drag Edge or Drag Plane command.**

c. Select BARREL VAULT from the Shapes pull-down on the Draw Model tool palette icon. Solid square handles will appear at the centroid of all visible planes.

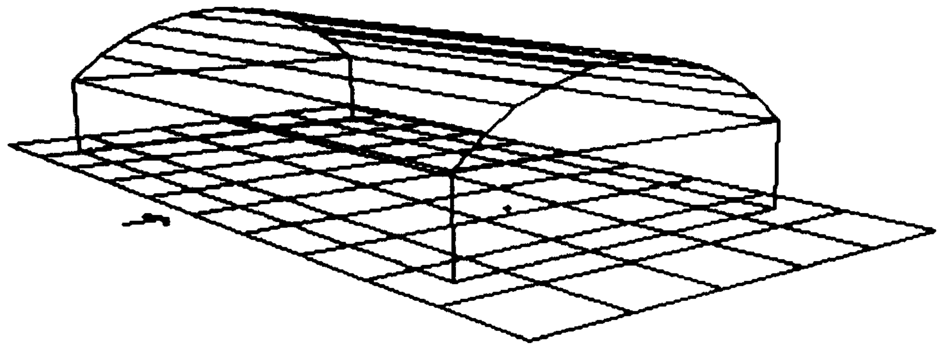
d. Select the top plane with the mouse. The barrel vault will automatically appear and assume the proportions of the plane selected to receive the barrel vault. The Dimensions pop-up dialog window will also appear with current dimensions inserted.

» **Note: The ridge of the barrel is used to define its orientation, which was set to E-W under Initial Shape Size.**

e. Click the left mouse key to fix the barrel vault shape to the selected plane. Solid square handles will reappear to stack another barrel vault on another plane if so desired.

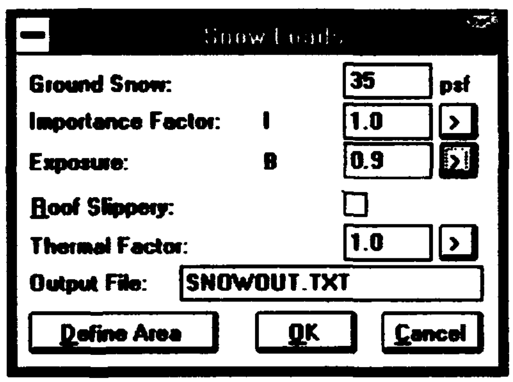
f. Double click the right mouse key to exit the Draw Shape on Plane command.

5. This completes the model for this example.



C. Develop snow loads on the roof

1. Select the LOADS AND DESIGN tool palette.
2. Select SNOW LOADS from the Loads pull-down menu or select the snow-flake icon. A Snow Loads pop-up dialog window will appear.



3. Verify parameters, modify as required, and select OK when satisfied. Roof snow load calculations will automatically begin.
4. The building plan and a section will appear upon completion of the snow load calculations.

D. Manipulation of the building model and its snow loads.

1. For details on the following abbreviated commands, refer to steps D-1 through D-7 in Snow Load Example 1.
   a. Zoom the graphics on the screen.
   b. Pan the screen image.
   c. View → Perspective 3D
   d. View → Solid Object
   e. Rotate 3D view.
   f. Adjust the viewing height.
   g. Adjust the viewing distance.

h. View → Show Loads.

✔ Unbalanced

OK

E. Generation of hard copies.

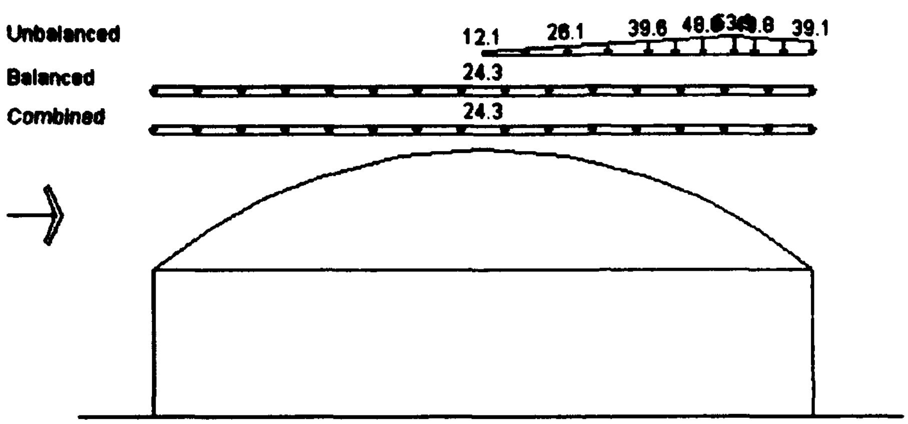
1. For details on the following abbreviated commands, refer to steps E-1 and E-2 of Snow Load Example 1.

   a. Viewpoint → Options → Section

   b. File → Print Screen

      ✔ Printer

      OK



Snow Loads (psf)

2. Review and print the snow load calculations. For details on the following abbreviated commands, refer to steps E-3 and E-4 of Snow Load Example 1.

   a. File → Print Data

      ✔ Snow

      ✔ Print to File

      ✔ Execute Notepad

      OK

   b. Notepad → File → Page Setup

      Left Margin : 0.5

      Right Margin : 0.0

      OK

   c. Notepad → File → Print

   d. Notepad → File → Exit

Example 2 sample output :

```
Project       : Theater
Location      : Milwaukee
Design Load   : TM 5-809-1 1986
Time          : Thu Sep 12, 1991   12:33 PM

********************** Arched Roof Snow Load Design ***********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Heated Structure.
Ct = 1.0
Importance Category: II
I  = 1.1
Pg = 35.0 psf
Pf = 24.25 psf
Roof Width   : 80.0 ft
Crown Height: 15.0 ft
Equivalent Slope Theta = 21 deg
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-----------------------+
|     Pf = 24.25 psf    |
+-----------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-----------------------+
|     Ps = 24.25 psf    |
+-----------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 21 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 41 deg
Use Case II

Crown
Punbal = 0.5*Ps
+-------------------------+
|   Punbal = 12.13 psf    |
+-------------------------+

30 deg point (30.4 ft from crown)
Punbal = 2*Ps/Ce
+-------------------------+
|   Punbal = 53.89 psf    |
+-------------------------+

Eave
Height of eave above grade or lower roof: 18.0 ft
Punbal = [2*Ps/Ce]*[1-(phi-30)/40]
+-------------------------+
|   Punbal = 39.07 psf    |
+-------------------------+
```

F. Save the building model with its snow loads applied for future reference.

1. **Save as filename:** TUTOR2.BLD.
2. **Refer to steps F-1 through F-4 in Example 1 for details on the following commands.**
   a. **File → Save**
      Enter filename
      OK

EXAMPLE THREE - [illegible] crown height of 5.0'

Example Two is repeated for a crown height of 5.0 feet to test the snow design requirements when the equivalent slope theta is less than 10 degrees and no unbalanced snow load is required.

A. Establish Criteria

1. Data will be reused from Example 2.

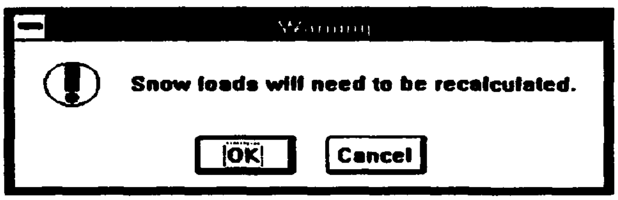
B. Draw Volumetric Model

1. Select OPEN from the File pull-down menu on the CASM menu bar.
   a. Select Filename: TUTOR2.BLD (file created in Example 2) from the scroll box files on the pop-up dialog window.
   b. Click the left mouse key on OK to load the building data. The barrel vaulted building will appear on the screen.

» **Note: If you were working on a different building, a warning pop-up box may appear to save that building's data before it loads the new data.**

2. Select DRAW MODEL from the CASM menu bar.

3. Delete the barrel vault shape.
   a. Select PERSPECTIVE 3D from the Options pull-down menu.
   b. Select DELETE SHAPE from the Edit pull-down menu. A warning pop-up box will appear stating that snow loads will need to be recalculated.



» **Note: Anytime snow or wind loads have been calculated, and then an attempt is made to alter the geometry, this warning box will appear.**

   c. Click the left mouse key on OK and solid square handles will appear at the centroids of the visible planes.

» **Note: More than one handle will appear for each constructed shape. Selecting any one of the handles for a given shape will delete the entire shape.**

   d. Select one of the barrel vault handles with the mouse and the shape is deleted. Handles will reappear to allow deletion of any remaining shapes.
   e. Double click on the right mouse key to exit the delete object command.

4. Insert a new barrel vault with a crown height of 5 feet.

a. Set the INITIAL SHAPE SIZE to 5 feet in height to reflect the new crown height.

b. Turn on STACK ON LAST SHAPE from the Layout pull-down menu.

c. Select BARREL VAULT from the Shapes pull-down menu. The barrel vault roof will appear on top of the last shape.

d. Click on the left mouse key to add the barrel vault.

» Note: You do not need to double click the right mouse key to exit the command, since you cannot stack another shape on top of a barrel vault.

5. This completes the model for this example.

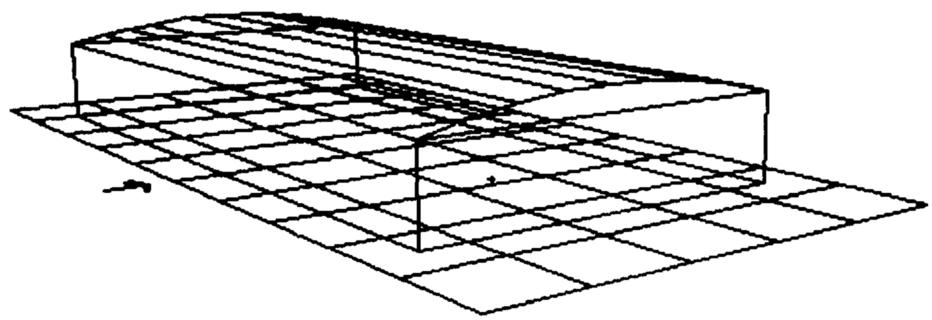
C. Develop snow loads on the roof.

1. For details on the following commands, refer to steps C-1 through C-4 of Snow Load Example 1.

a. Loads and Design → Loads → Snow

Review values

OK



D. Manipulation of building model and its snow load.

1. For details on the following abbreviated commands, refer to steps D-1 through D-7 in Snow Load Example 1.

a. Zoom the graphics on the screen.

b. Pan the screen image.

c. View → Perspective 3D

d. View → Solid Object

e. Rotate 3D view.

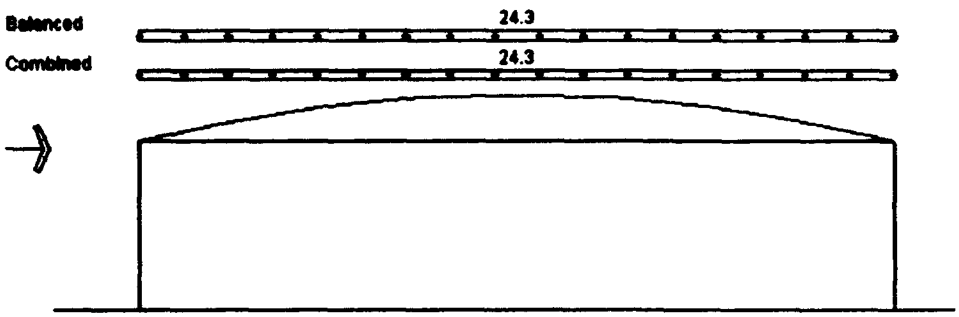
f. Adjust the viewing height.

g. Adjust the viewing distance.

h. View → Show Loads.

✔ Unbalanced

OK



Snow Loads (psf)

E. Generation of hard copies.

1. Print a 2-D section and calculations. For details on the following abbreviated commands, refer to steps E-1 through E-4 of Snow Load Example 1.
   a. Viewpoint → Options → Section
   b. File → Print Screen
      ✔ Printer
      OK
   c. File → Print Data
      ✔ Snow
      ✔ Print to File
      ✔ Execute Notepad
      OK
   d. Notepad → File → Page Setup
      Left Margin : 0.5
      Right Margin : 0.0
      OK
   e. Notepad → File → Print
   f. Notepad → File → Exit

Example 3 Sample Output :

```
Project     : Theater
Location    : Milwaukee
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991  12:33 PM

********************** Arched Roof Snow Load Design ***********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Heated Structure.
Ct = 1.0
Importance Category: II
I  = 1.1
Pg = 35.0 psf
Pf = 24.25 psf
Roof Width   : 80.0 ft
Crown Height: 5.0 ft
Equivalent Slope Theta = 7 deg
Check minimum Pf where theta <= 10 deg
When Pg > 20.0 psf, min Pf = 20*I
Min Pf = 22.00 psf
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-------------------------+
|      Pf = 24.25 psf     |
+-------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-------------------------+
|      Ps = 24.25 psf     |
+-------------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 7 deg
Theta < 10 deg or > 60 deg, unbalanced condition does not apply.
```

F. Save the building model with its snow loads applied for future reference.

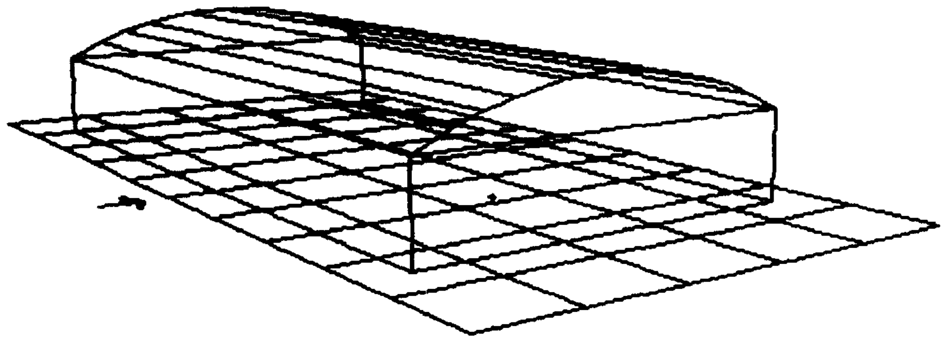
1. Select SAVE AS from the File pull-down menu on the CASM menu bar. This allows us to save the building data in a different file from Example 2.

» **Note: Selecting SAVE would have replaced the existing file without allowing you to change the filename.**

2. Type in Filename: TUTOR3.BLD.
3. Click mouse on OK.

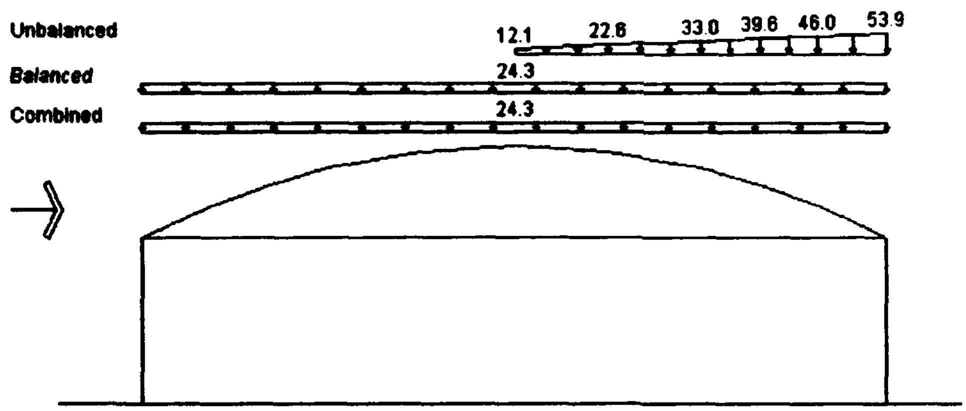
EXAMPLE FOUR: Arched roof crown height of 10'-0"

Example Two is repeated for a crown height of 10 feet to test the snow design requirements for a CASE I situation that does include an unbalanced snow load.



<u>Repeat all the steps of Example 3, but revise the crown height to 10 feet.</u>

<u>Example 4 sample output :</u>



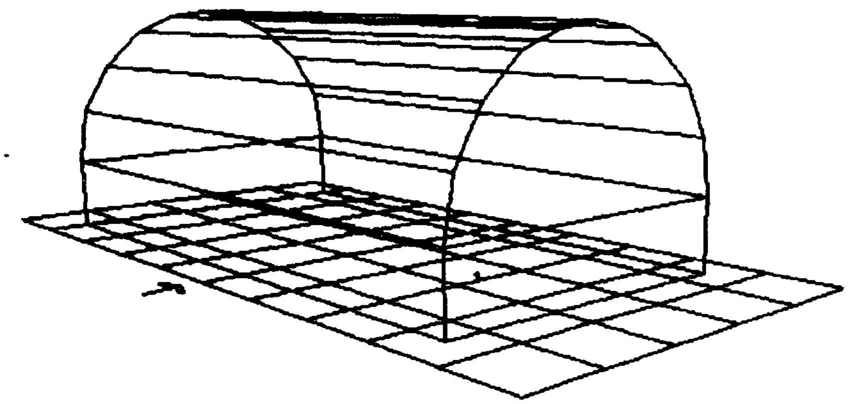
Snow Loads (psf)

```
Project      : Theater
Location     : Milwaukee
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   12:34 PM
*********************** Arched Roof Snow Load Design ***********************
Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Heated Structure.
Ct = 1.0
Importance Category: II
I  = 1.1
Pg = 35.0 psf
Pf = 24.25 psf
```

```
Roof Width  : 80.0 ft
Crown Height: 10.0 ft
Equivalent Slope Theta = 14 deg
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-----------------------+
|     Pf = 24.25 psf    |
+-----------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-----------------------+
|     Ps = 24.25 psf    |
+-----------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 14 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 28 deg
Use Case I

Crown
Punbal = 0.5*Ps
+-----------------------+
|  Punbal = 12.13 psf   |
+-----------------------+

Eave
Punbal = 2*Ps/Ce
+-----------------------+
|  Punbal = 53.89 psf   |
+-----------------------+
```

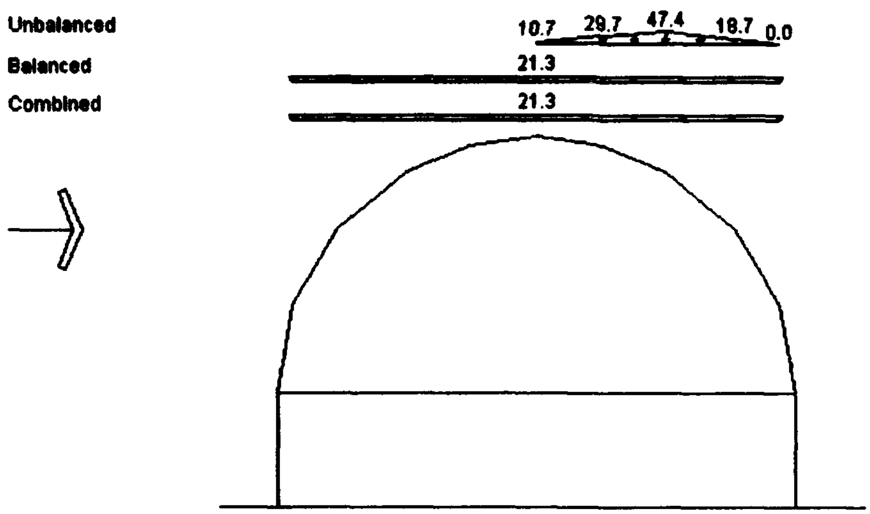
[illegible]

Example Two is repeated for a crown height of 40 feet to test the snow design requirements for a CASE III situation where the 'eave' is now located at the phi = 70° location of 38 feet from the crown.



Repeat all the steps of Example 3, but revise the crown height to 40 feet.

Example 5 sample output :



Snow Loads (psf)

```
Project      : Theater
Location     : Milwaukee
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   12:35 PM

*********************** Arched Roof Snow Load Design ************************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
```

```
Heated Structure.
Ct = 1.0
Importance Category: II
I  = 1.1
Pg = 35.0 psf
Pf = 24.25 psf
Roof Width  : 80.0 ft
Crown Height: 40.0 ft
Equivalent Slope Theta = 35 deg
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+------------------------+
|     Pf = 24.25 psf     |
+------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 0.88
+------------------------+
|     Ps = 21.34 psf     |
+------------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 35 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 90 deg
Use Case III

Crown
Punbal = 0.5*Ps
+------------------------+
|   Punbal = 10.67 psf   |
+------------------------+

30 deg point (20.0 ft from crown)
Punbal = 2*Ps/Ce
+------------------------+
|   Punbal = 47.42 psf   |
+------------------------+

Eave
Height of eave above grade or lower roof: 18.0 ft
70 deg point (37.6 ft from crown)
+------------------------+
|   Punbal = 0.00 psf    |
+------------------------+
```

[illegible] Height of 15'-0"

Example Two is repeated for the original crown height of 15 feet, but with a zero height of eave above grade. The arch thus originates from grade. This is to illustrate the alternate distribution of snow accumulation when a lower roof or grade exists within 3 feet of the eave.

This example could begin as a new building; however,it will be approached by opening the saved Example 2. This eliminates reentering the Design Criteria.

A. Establish Criteria.

1. Already entered in Example 2.

B. Draw Volumetric model.




1. OPEN Filename: TUTOR2.BLD (file was created in Example 2).
2. Select DRAW MODEL tool palette.
3. Delete both existing shapes.
   a. Select PERSPECTIVE 3D from the Options pull-down menu.
   b. Select DELETE SHAPE from Edit pull-down menu.
   c. Select OK on the warning pop-up box to indicate that snow loads will need to be recalculated.
   d. Select one handle on the barrel vault shape.
   e. Select one handle on the cube shape.

» **Note: You will not need to double click the right mouse key, since there are no more objects to delete.**


4. Draw the new barrel vault on the ground plane.
   a. Turn on STACK ON GROUND.
   b. Select BARREL VAULT.

   Oops! The object size does not reflect the required proportions for this example. The editing functions cannot be used to completely revise the barrel vault size.

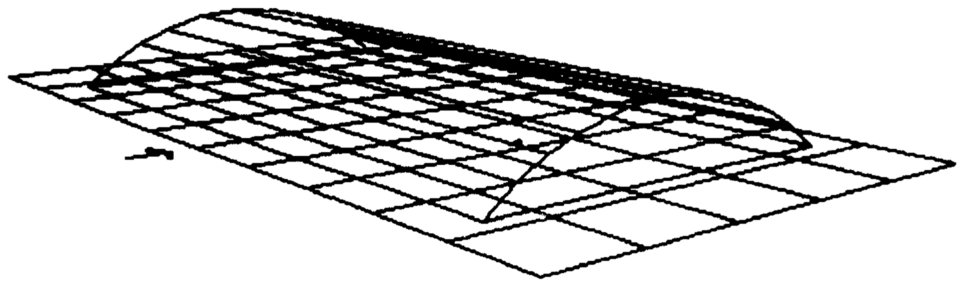
   c. Double click the right mouse key to exit the command and not add the barrel vault.

   d. Set the INITIAL SHAPE SIZE as follows:

WIDTH N-S	:80 feet
E-W	:150 feet
HEIGHT	:15 feet
ORIENTATION	:E-W

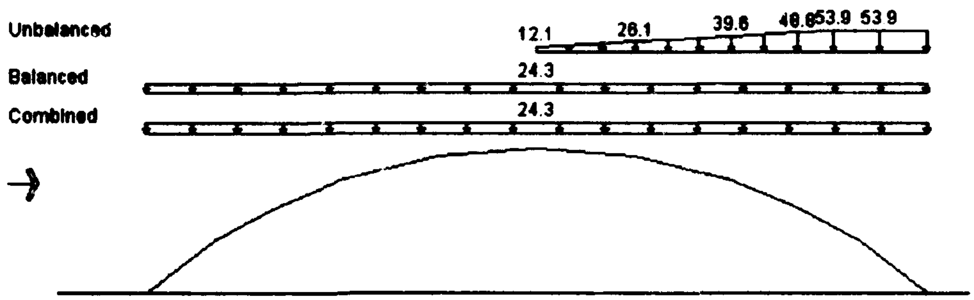
   e. Select BARREL VAULT again.
   f. Click the left mouse key to add the shape.
   g. Double click the right mouse key to stop adding barrel vaults to the ground plane.

5. **This completes the model for this example.**



C. Develop snow loads on the roof.

D. Manipulate the building model and its snow load.



Snow Loads (psf)

E. Generate hard copies.

F. Save the building model with its snow loads applied for future use.

Example 6 sample output :

```
Project      : Theater
Location     : Milwaukee
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   12:36 PM

********************** Arched Roof Snow Load Design **********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Heated Structure.
Ct = 1.0
Importance Category: II
I  = 1.1
Pg = 35.0 psf
Pf = 24.25 psf
Roof Width   : 80.0 ft
Crown Height: 15.0 ft
Equivalent Slope Theta = 21 deg
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
```

```
+-----------------------+
|    Pf = 24.25 psf     |
+-----------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-----------------------+
|    Ps = 24.25 psf     |
+-----------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 21 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 41 deg
Use Case II

Crown
Punbal = 0.5*Ps
+-----------------------+
|  Punbal = 12.13 psf   |
+-----------------------+

30 deg point (30.4 ft from crown)
Punbal = 2*Ps/Ce
+-----------------------+
|  Punbal = 53.89 psf   |
+-----------------------+

Eave
Height of eave above grade or lower roof: 0.0 ft
Eave <= 3 ft above grade or lower roof.
+-----------------------+
|  Punbal = 53.89 psf   |
+-----------------------+
```

■ DRIFTED AND SLIDING SNOW

Drifts may occur on lower roofs sited within 20 feet of a higher adjacent structure and also from projections above a lower roof. These projections may be parapets, penthouses, stair and elevator projections, mechanical equipment, etc. The snow load algorithm searches for the drift criteria stated above directly from the building model.

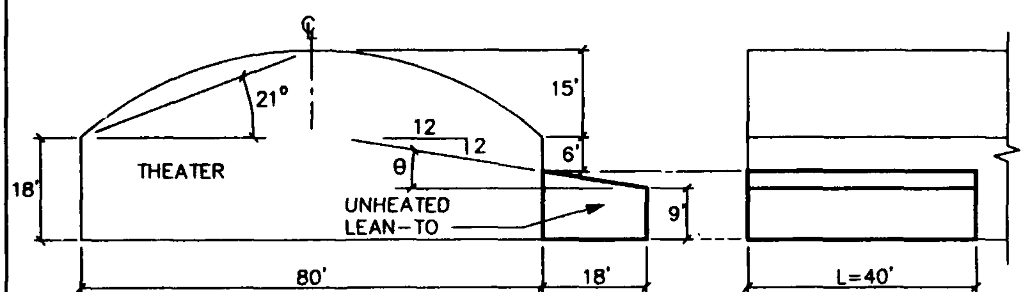
Sliding snow occurs on roofs situated below gable or shed roofs having a slope (or equivalent slope in the case of barrel vaults) greater than or equal to 2 in 12.

Lower roofs which are located below roofs having a slope (or equivalent slope) greater than or equal to 2 in 12 shall include sliding effects per the criteria detailed by the Metal Building Manufacturer's Association (MBMA) document, Low Rise Building Systems Manual, 1986. The following example illustrates the procedure.

EXAMPLE SEVEN: Lean-to roof adjacent to taller roof

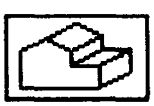
Dimensions, siting, and location are the same as Example 2. The addition is unheated, but the theater is heated. This example is predominantly taken from TM 5-809-1 1986, page E-6.

Given: A lean-to roof structure for a storage and office addition adjacent to a taller theater roof.



Required: Find the design snow load on the lean-to roof, including drift and sliding considerations.

Solution:


A. Establish Criteria.

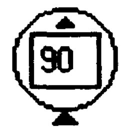
1. Open Filename: TUTOR2.BLD (file was created in Example 2).

B. Draw volumetric model.

1. Select DRAW MODEL.
   a. Select PERSPECTIVE 3D from the Options pull-down menu.

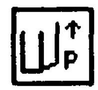
2. Oops! The ground plane is of insufficient size to add the shed in this example if the building is centered on the ground plane width.
   a. Increase the Ground Plane N-S WIDTH to 140 feet.
3. Add the shed.
   a. Turn on STACK ON PLANE .

   b. Rotate the 3-D Viewpoint of the building so you are looking from the SE quadrant.



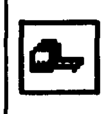
   c. Select CUBE and handles will appear.

   d. Select south wall handle and the cube will expand to match the wall dimensions.
   e. Click the left mouse key to fix the shed.
   f. Double click the right mouse key to exit Stack On Plane command.
   g. Proportion the shed.
      (1) Drag the shed roof plane down 6 feet from the barrel vault eave using the drag plane command.



>> **Note: The height shown in the Dimensions pop-up dialog box will be 12 feet.**

      (2) Drag the east plane of the shed to the west and make the length of the shed 40 feet.
      (3) Drag the south plane of the shed to make its width 18 feet.
      (4) Double click the right mouse key to exit drag plane.
   h. Create lean-to roof.
      (1) Select TAPE MEASURE from the Edit pull-down menu.



>> **Note: Roof slopes shown in the Dimensions dialog window may not be correct for a Cube. This issue can be avoided by checking roof slopes with TAPE MEASURE.**

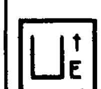
      (2) Position the mouse pointer at the NE upper vertex of the shed. Click the left mouse key. A red dot will appear at the vertex.
      (3) Position the mouse pointer at the SE upper vertex of the shed. Click the left mouse key. A red dot will appear at the vertex, and a dashed red line will connect the two dots. A pop-up Measure dialog window will appear displaying data regarding the dashed line between those two red dots.

>> **Note: You cannot edit the data in this dialog window. The values in the data blocks will change as you drag an edge which is connected to the vertices.**

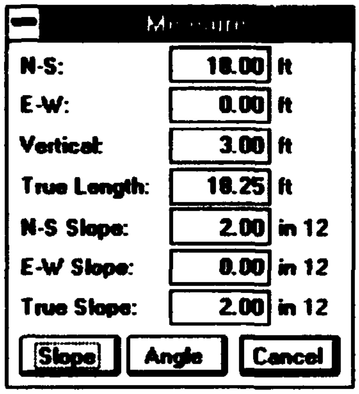
>> **Note: On the single-screen CASM version you may need to move the Measure dialog window to a more convenient location on the screen.**

      (4) LOCK the N-S direction.

      (5) Select DRAG EDGE and handles will appear.



(6) Select the upper south edge of the shed. Hold the right mouse key down while moving it up and down. Drag the edge vertically until the N-S slope in the Measure dialog window reads 2.00 in 12.



(7) Click the left mouse key to fix the roof slope.

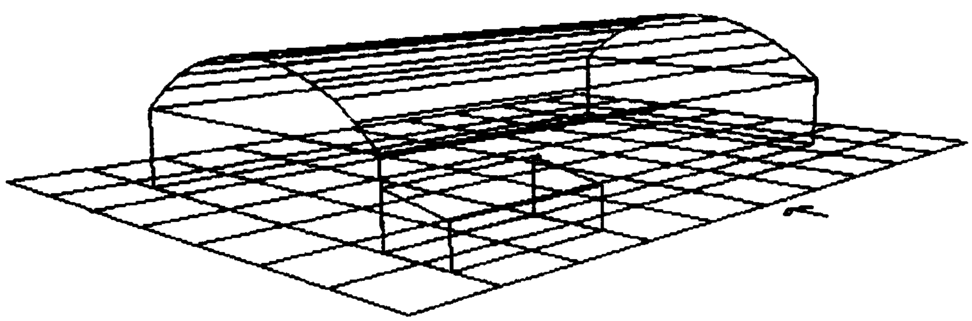
(8) Double click the right mouse key to exit the Drag Edge command.

(9) Select CANCEL on the Measure dialog window to stop tape measuring.

» Note: Tape Measure remains active until it is canceled.

(10) UNLOCK the N-S direction.

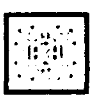
4. This completes the model for this example.



C. Develop snow loads on the shed roof.

1. Select LOADS AND DESIGN .

2. Select SNOW LOADS . A pop-up dialog window will appear.



a. Select the THERMAL FACTOR for an Unheated roof.

» Note: The barrel vault portion of the building is heated, while the shed is unheated. The user must decide for which portion of the total building results are required. For this example, the shed roof is under consideration; thus, "unheated" was chosen. The barrel vault snow loads will therefore not be correct. A separate snow load study is

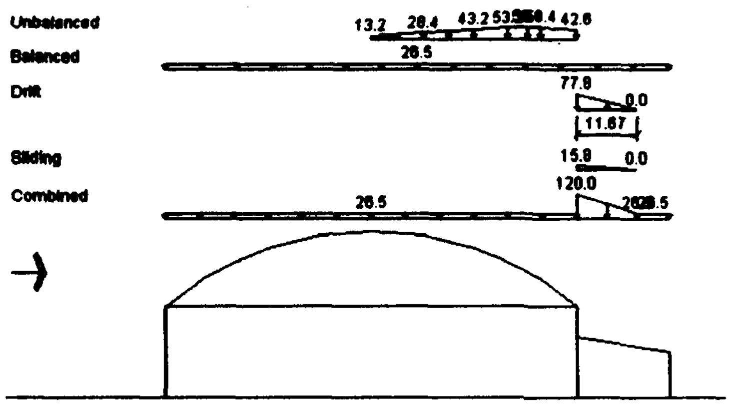
required for the barrel vault, which will then make the shed output incorrect.

b. **Select Importance Category I, since the shed has an occupancy less than 300 people.**

c. **Select OK when satisfied with all the parameters. Roof snow load calculations will automatically begin. A pop-up dialog box will keep you informed of the program's progress.**

d. **The building plan and a section will appear when the calculations are complete. Note that drift and sliding snow values exist over the shed portion of the roof.**

D. Manipulate the building model and its snow load.



Snow Loads (psf)

E. Generate hard copies.

F. Save the building model with its snow loads applied for future use.

Example 7 sample output :

```
Project      : Theater
Location     : Milwaukee
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   12:37 PM

********************** Arched Roof Snow Load Design **********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Unheated Structure.
Ct = 1.2
Importance Category: I
I  = 1.0
Pg = 35.0 psf
Pf = 26.46 psf
Roof Width   : 80.0 ft
Crown Height: 15.0 ft
Equivalent Slope Theta = 21 deg
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+------------------------+
|     Pf = 26.46 psf     |
+------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+------------------------+
|     Ps = 26.46 psf     |
+------------------------+
```

Invalid for barrel vault roof portion.

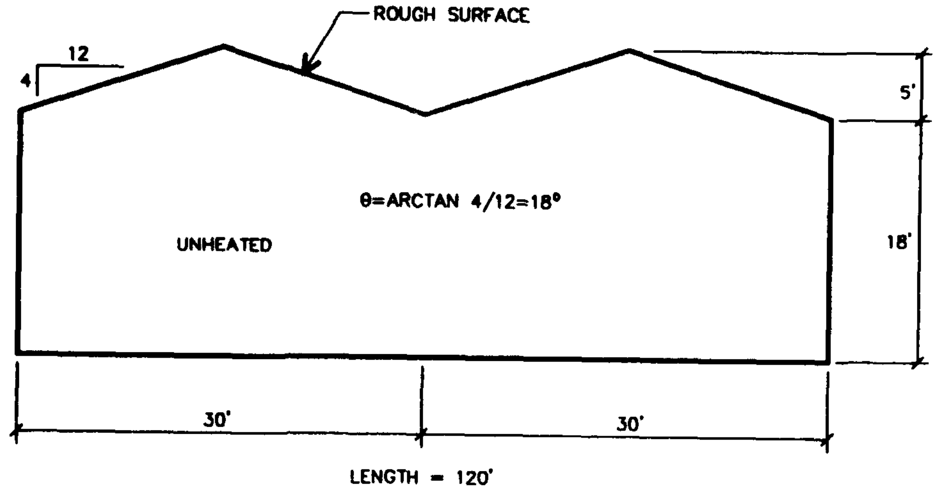
```
Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 21 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 41 deg
Use Case II

Crown
Punbal = 0.5*Ps
+-------------------------+
|   Punbal = 13.23 psf    |
+-------------------------+

30 deg point (30.4 ft from crown)
Punbal = 2*Ps/Ce
+-------------------------+
|   Punbal = 58.80 psf    |
+-------------------------+

Eave
Height of eave above grade or lower roof: 18.0 ft
Punbal = [2*Ps/Ce]*[1-(phi-30)/40]
+-------------------------+
|   Punbal = 42.63 psf    |
+-------------------------+
********************** Arched Roof Snow Load Design **********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Unheated Structure.
Ct = 1.2
Importance Category: I
I  = 1.0
Pg = 35.0 psf
Pf = 26.46 psf
Roof Width  : 80.0 ft
Crown Height: 15.0 ft
Equivalent Slope Thet[illegible]
Since theta > 10 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-------------------------+
|     Pf = 26.46 psf      |
+-------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-------------------------+
|     Ps = 26.46 psf      |
+-------------------------+

Unbalanced Snow Load (Punbal)
Since equivalent slope, theta, is 21 deg
10 deg < theta < 60 deg, unbalanced condition applies.
Where slope at eaves = 41 deg
Use Case II

Crown
Punbal = 0.5*Ps
+-------------------------+
|   Punbal = 13.23 psf    |
+-------------------------+

30 deg point (30.4 ft from crown)
Punbal = 2*Ps/Ce
+-------------------------+
|   Punbal = 58.80 psf    |
+-------------------------+

Eave
Height of eave above grade or lower roof: 6.0 ft
Punbal = [2*Ps/Ce]*[1-(phi-30)/40]
+-------------------------+
|   Punbal = 42.63 psf    |
+-------------------------+
******************* Flat/Lean-To Roof Snow Load Design ********************

Flat Roof Snow Load (Pf)
Pf = 0.7*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Unheated Structure.
Ct = 1.2
Importance Category: I
I  = 1.0
Pg = 35.0 psf
Pf = 26.46 psf
Roof Slope: 2.00 in 12
```

Invalid for barrel vault roof portion.

```
Theta = 9 deg
Check minimum Pf where theta <= 15 deg
When Pg > 20.0 psf, min Pf = 20*I
Min Pf = 20.00 psf
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+------------------------+
|     Pf = 26.46 psf     |
+------------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+------------------------+
|     Ps = 26.46 psf     |
+------------------------+

************************* Drift Snow Load Design **************************

Pg = 35.0 psf
Snow Density = 20.0 pcf
Ps = 26.5 psf (rain-on-snow surcharge not included)
hb = Ps/density
hb = 1.32 ft
Projection Height = 6.00 ft
hc = height-hb
hc = 4.68 ft
hc/hb = 3.54 >= 0.20  Therefore consider drift load.
Importance Category: I
I  = 1.0
Snow Exposure Category: B
Ce = 0.9
Separation = 0.00 ft
hd = 2*I*Pg/Ce*density*(20-s/20) <= hc
hd = 3.89 ft
hd <= hc
Pd = hd*density
+------------------------+
|     Pd = 77.78 psf     |
+------------------------+

Width of drift for L = 40.00 <= 50 ft: W = 3*hd >= 10 ft.
+------------------------+
|     W = 11.67 ft       |
+------------------------+

************************ Sliding Snow Load Design *************************

Theta = 21 deg > 2 in 12, therefore sliding snow.
Projection Height (hr) = 6.00 ft
Separation = 0.00 ft
Separation distance < hr and < 20 ft
Increase in drift height: hs = 0.4*hd
hs = 1.56 ft
hd + hs <= hc
hd + hs = 5.45 ft
hc = 4.68 ft
Height used = 4.68 ft
Pd + Ps = height*density
+------------------------+
|  Pd + Ps = 93.54 psf   |
+------------------------+

Notes for sliding snow:
   Calculations based on MBMA 1986.
```

EXAMPLE [illegible]: Multiple-gable roof

Given: This multiple-gable roof example is taken from page E-2 of TM 5-809-1 1986. It is a warehouse located in Anchorage, Alaska, and the site is a windy field with a few birch trees planted nearby. It is an unheated structure with roofing that creates a rough surface. The following dimensional data is given, and there are no adjacent structures.



Required: Determine the balanced and unbalanced snow loads.

Solution:

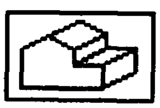
A. Establish Criteria.

1. Input the following data into the PROJECT, REGIONAL, and SITE Criteria dialog windows:

PROJECT:	Project Name	: Warehouse
	City/Installation	: Anchorage
	State	: AK
	Design Load	:TM 5-809-1 1986
REGIONAL:	Ground Snow Load	: 45 psf
SITE SNOW:	Importance	: Category I
	Exposure	: Category B
	Roof Slippery	: no
	Thermal Factor	: Unheated

» **Note: The CASM program uses the abbreviations for the names of states rather than the full name. The insertion of AK for Alaska switches the program from TM equation 6-1a to 6-1b.**

B. Draw volumetric model



1. Select DRAW MODEL from the CASM menu bar.
2. Establish general layout requirements, which are different than previously established.

a. Use the following:

DEFINE UNITS (snap increment)		: [illegible] inches
SNAP TO UNITS		: on
SHOW GROUND PLANE		: on
GROUND PLANE		
WIDTH	N-S	: 140 feet
	E-W	: 140 feet
SPACING	N-S	: 20 feet
	E-W	: 20 feet
INITIAL SHAPE SIZE		
N-S WIDTH		: 120 feet
E-W WIDTH		: 60 feet
HEIGHT		: 18 feet
ORIENTATION		: N-S
STACK ON GROUND PLANE		: on
DIRECTIONS LOCKED		: none

1. Place a CUBE on the ground plane with the required dimensions.

2. Draw the multiple-gable roof.

   a. Change the following INITIAL SHAPE SIZE values:

E-W WIDTH	: 30.0 feet
HEIGHT	: 5.0 feet
MAINTAIN INITIAL SIZE	: on

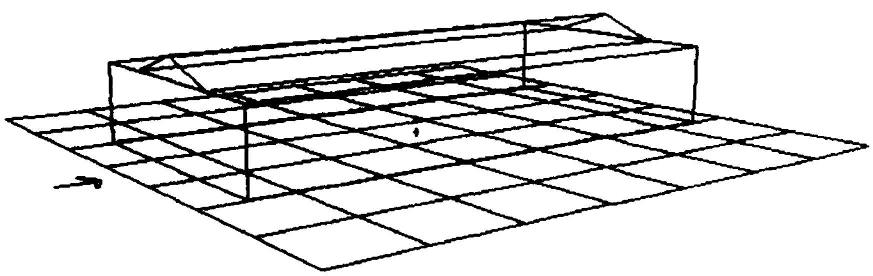
» **Note: If Maintain Initial Size in ON, the proportions of the shape when stacked will not be adjusted to the size of the plane.**

   b. Turn on STACK ON PLANE.

   c. Add one half of the multiple-gable roof.

   (1) Select PRISM from the Shapes pull-down menu. Handles will appear on all the visible planes.

   (2) Select the top plane of the cube with the mouse pointer. The plane is highlighted and a prism will appear at the Initial Shape Size of 30.0 feet wide, 120.0 feet long, and 5.0 feet high at the center of the plane.



   (3) Move the mouse left and right to position the prism at one edge of the cube.

   (4) Click the left mouse key to fix the position of the prism. Handles will appear on planes to stack another prism onto.

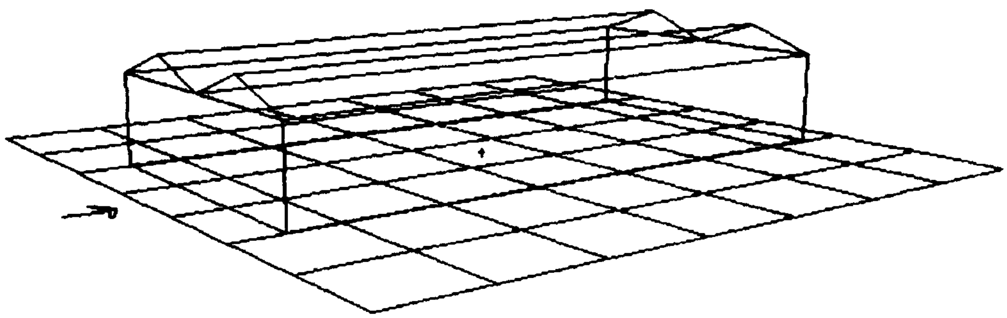
(5) **Select the top plane of the cube with the mouse pointer. The plane is highlighted, and a prism will appear on top of the cube.**

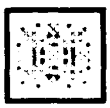
(6) **Move the mouse left and right to position the prism at the other edge on the cube.**

(7) **Click the left mouse key to fix the position of the prism. Handles will appear on planes to stack another prism onto.**

(8) **Double click the right mouse key to stop stacking prisms on planes.**

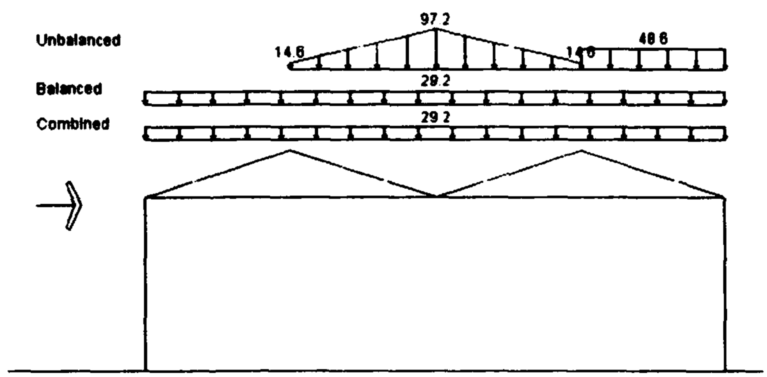
3. **This completes creation of the model.**





A. Develop snow loads on the roof.

B. Manipulate the building model and its snow load.



Snow Loads (psf)

C. Generate hard copies.

D. Save the building model with its snow load applied for future use.

Example 8 sample output :

```
Project      : Warehouse
Location     : Anchorage
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   1:17 PM

*************** Multiple Folded Plate Roof Snow Load Design ***************

Flat Roof Snow Load (Pf)
Pf = 0.6*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
```

```
Unheated Structure.
Ct = 1.2
Importance Category: I
I  = 1.0
Pg = 45.0 psf
Pf = 29.16 psf
Roof Slope: 4.00 in 12
Theta = 18 deg
Since theta > 15 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-----------------------+
|     Pf = 29.16 psf    |
+-----------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-----------------------+
|     Ps = 29.16 psf    |
+-----------------------+

Note: See Gable output for first windward and last leeward slope.

Unbalanced Snow Load (Punbal)

Ridge
Punbal = 0.5*Pf
+-------------------------+
|   Punbal = 14.58 psf    |
+-------------------------+

Valley
Punbal = 3*Pf/Ce
Punbal = 97.20 psf
Height of unbalanced load = 4.86 ft <= height of ridge = 5.00 ft
+-------------------------+
|   Punbal = 97.20 psf    |
+-------------------------+

********************* Gable/Hip Roof Snow Load Design *********************

Flat Roof Snow Load (Pf)
Pf = 0.6*Ce*Ct*I*Pg
Snow Exposure Category: B
Ce = 0.9
Unheated Structure.
Ct = 1.2
Importance Category: I
I  = 1.0
Pg = 45.0 psf
Pf = 29.16 psf
Roof Slope: 4.00 in 12
Theta = 18 deg
Since theta > 15 deg, min. snow load does not apply.
Since theta > 1/2 in/ft, rain-on-snow surcharge does not apply.
+-----------------------+
|     Pf = 29.16 psf    |
+-----------------------+

Sloped Roof Snow Load (Ps)
Ps = Cs*Pf
Roof Slippery: No
Cs = 1.00
+-----------------------+
|     Ps = 29.16 psf    |
+-----------------------+

Unbalanced Snow Load (Punbal)
Since 15 deg < theta < 70 deg, unbalanced condition applies.
Punbal = 1.5*Ps/Ce
+-------------------------+
|   Punbal = 48.60 psf    |
+-------------------------+
```

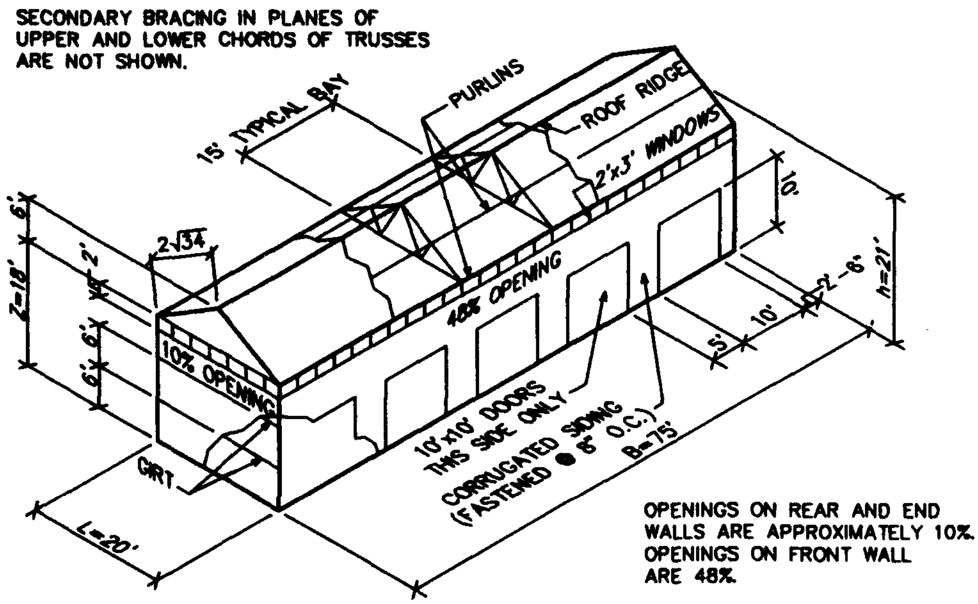
WIND LOADS

This section includes examples of wind load design for (1) main wind force resisting systems, (2) building components and cladding, and (3) unenclosed buildings. Subtopics under each of these three categories will address various roof forms and building heights to illustrate all provisions of the Tri-Services Load Assumption for Buildings Technical Manual, 1986.

■ Main Wind Force Resisting Systems

[illegible]

Given: This one-story gabled roof industrial building is an example taken from page D-1 of TM 5-809-1 1986. It is to be used for storage and maintenance of equipment. It is located in Huntsville, AL and is sited in exposure category C. See illustration below:



Required: Determine the external pressures and suctions on all surfaces for wind perpendicular and parallel to the ridge.

Solution:

A. Establish Criteria.

1. Select CRITERIA from the CASM menu bar and scroll down the pull-down menu to PROJECT.
2. Insert the following data:

info

Project name:	Industrial building
City/installation:	Huntsville
State:	AL
Design load:	TM5-809-1 1986

» **Note: No database exists for Huntsville and you will have to fill in any other useful data. The elevation above sea level is not used in this example, and a default factor of 0.00256 is assumed to reflect air mass density for the so-called "standard atmosphere" in calculations of velocity pressure q_z. Select OK when finished inputting data.**

3. Select CRITERIA for a second time and scroll down the pull-down menu to REGIONAL.
4. Insert the following data into the appropriate boxes within the REGIONAL WINDOW:

Basic Wind Speed :	70 mph (from the basic wind speed map)
Coastal :	no (leave the box blank)

» **Note: It is important not to omit consideration of coastal. If you click on the box, an X will appear, indicating that you are within 100 miles of the coastline. You will be asked for the distance later. Should you leave the box blank, it is assumed that you are inland and 100 miles will be the default value used later.**

5. Select OK when finished, since no other data is required here.
6. Again select CRITERIA and scroll down the pull-down menu to SITE.
7. Insert the following WIND data into the appropriate boxes within the SITE window:

Importance :	1.0
Exposure :	C

The pop-up dialog windows will assist you in the selections above.

» **Distance to oceanline: If the coastal box was left blank, 100 miles will exist here. This value cannot now be changed. If coastal was checked X, you may enter any number less than 100 miles.**

8. Select OK when you have finished entering data. The CASM program will return to the CASM program window.

B. Draw volumetric model.
   1. Select the DRAW MODEL tool palette.
   2. Establish general layout requirements for this example.
      a. Use the following:

DEFINE UNITS(snap increment)		: 12 inches
SNAP TO UNITS		: on
SHOW GROUND PLANE		: on
GROUND PLANE		
WIDTH	N-S	: 100 feet
	E-W	: 100 feet
SPACING	N-S	: 20 feet
	E-W	: 20 feet
INITIAL SHAPE SIZE		
N-S WIDTH		: 20 feet
E-W WIDTH		: 75 feet
HEIGHT		: 18 feet
ORIENTATION		: E-W
STACK ON GROUND PLANE		: on
DIRECTIONS LOCKED		: none

3. Create the first floor building volume.


   a. Select CUBE and center on the ground plane.

4. Create the gable roof form.

   a. Turn on STACK ON LAST SHAPE from Layout pull-down menu.

   b. Select PRISM .

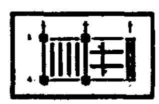
   c. LOCK the N-S and E-W directions.

   d. Select DRAG EDGE from Edit pull-down menu. Lower the ridge to 6 feet in height by selecting the handle on the ridge with the mouse pointer and by holding down the right mouse key while dragging the mouse toward you.

   e. UNLOCK the N-S and E-W directions.

5. Create wall openings.

>> **Note: It is not necessary to draw all the wall openings on every elevation, if you already know the internal pressure coefficients (GCpi) for your model. The computer will automatically compute the required GCpi if you are uncertain as to whether you meet the criteria upon which GCpi values are based. We will let the computer do the calculations for us in this example.**

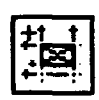


   a. Select the DRAW STRUCTURE tool palette.

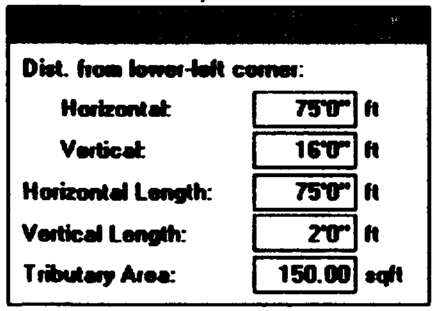
   b. Select VERTICAL STRUCTURAL PLANE from the View pull-down menu. Handles will appear on the visible vertical planes.

   c. Select the south plane with the mouse and a 2-D elevation of the selected wall will appear.

   d. Create the continuous top window opening.



      (1) Select ADD OPENING from the Grid/Open pull-down menu. A Tributary Area dialog window will appear which shows the distance of the mouse pointer from the lower left corner of the 2-D view, lengths of the opening, and the tributary area of the opening.



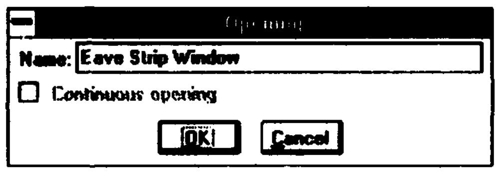
(2) Locate the mouse pointer at the upper left corner of the elevation. Single click the left mouse key to fix one opening corner.

» **Note: If you are having trouble positioning the pointer with the mouse, use the arrow keys instead. The [Enter] key is the same as single clicking the left mouse key.**

(3) Drag the mouse to dynamically proportion the opening until the vertical length is 2 feet and the horizontal length is 75 feet.

(4) Click the left mouse key when the desired opening size is achieved.

(5) A dialog box appears asking you to name the opening. Type name: Eave Strip Window.



» **Note: The CONTINUOUS openings option only applies to horizontal floor planes. It would duplicate the opening on all other floors in the model.**

(6) Click on OK when finished to add the opening.

» **Note: Clicking on CANCEL will not insert the opening.**

e. Create the overhead door openings.

(1) Set the SNAP INCREMENT to 6 inches from the Define Units command by selecting the Define Units icon button in the lower right corner of the screen.

(2) Select ADD OPENING from the Grid/Open pull-down menu. The Tributary Area pop-up dialog window will again appear.

(3) Move the mouse pointer until the distance from the lower left corner is 2.5 feet horizontally and 0.0 feet vertically. Single click the left mouse key to fix the corner of the opening.

(4) Drag the mouse until the door proportions are 10 feet horizontally and vertically.

(5) Click the left mouse key to fix the opening dimensions.

(6) Name the opening: Door 1.

(7) Click on OK to add the opening.

(8) Repeat steps (4) through (9) to add the remaining openings on this elevation.

(9) If you selected incorrect coordinates for an opening, use the Delete Opening or Modify Opening commands. See the Reference guide for details on using these commands.

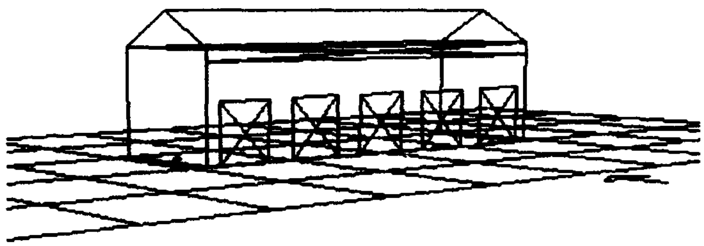
f. Select PERSPECTIVE (3D) from the Viewpoint Options pull-down menu to view the model at this stage.

g. Add eave strip windows along the other three elevations. Follow steps 5b through 5d.

» Note: You will have to select PERSPECTIVE (3D) and rotate the model to view other elevations to facilitate wall selection.

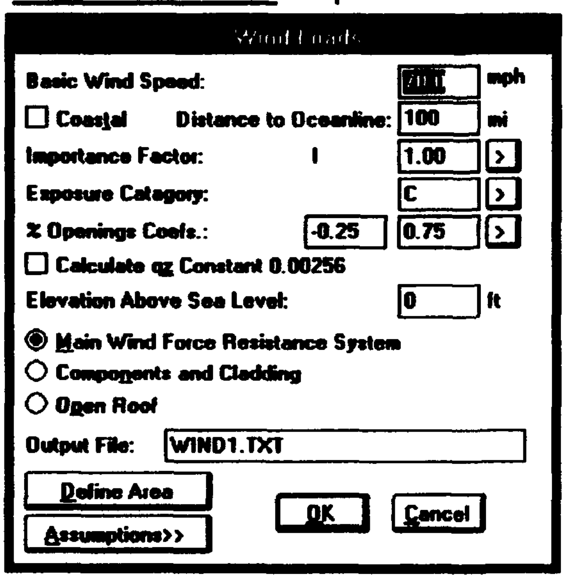
» Note: Only openings from the last selected plane will be shown on the 3-D model. To see all openings on all elevations, select SHOW STRUCTURE and select the check box for ALL PLANES.

h. This completes the model and its openings for this example.

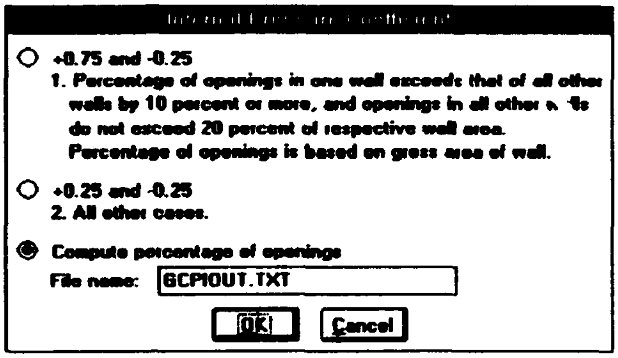


C. Develop Main Wind Force Resistance wind loads on the building.

1. Select the LOADS AND DESIGN tool palette.



2. Select WIND from the Loads pull-down menu. A Wind Loads pop-up dialog window will appear with values selected under Criteria.
3. Select % OPENINGS COEFS. by clicking the data window button. An Internal Pressure Coefficients pop-up dialog window will appear listing options and their criteria.



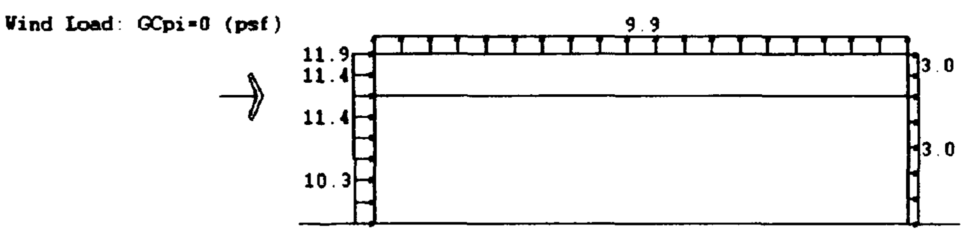
   a. Select COMPUTE PERCENTAGE OF OPENINGS to have the computer select GCpi based on code criteria.
   b. Modify filename if desired.
   c. Select OK and internal pressure criteria will be tested. A pop-up dialog window will appear to keep you informed of the calculation progress. Final GCpi values will be displayed in the Wind Load dialog window.

» **Note: The output file can be displayed and printed similar to snow load calculations.**

4. Turn Off CALCULATE QZ CONSTANT 0.00256.
5. Turn on MAIN WIND FORCE RESISTANCE SYSTEM.
6. Modify Output Filename as desired.

» **Note: Selection of ASSUMPTIONS allows the user to choose a plan and height ratio for appropriate consideration of B/L and h/L ratios for irregular building forms. An option to use the eave height, rather than the roof mean height, for slopes less than 10 degrees can also be selected. See the Reference Manual for further elaboration.**

7. Click on OK when satisfied, and wind load calculations will begin. A pop-up dialog window will keep you informed of the progress.

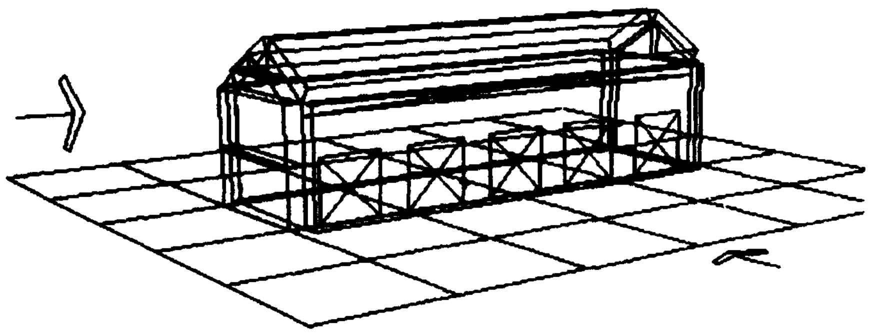


8. The building plan and a section will appear upon completion of the calculations. Wind pressures and suctions will be shown for the wind direction arrow displayed and with GCpi = 0.

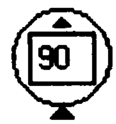
D. Manipulation of the building model and its wind loads.

1. **Change wind direction.**

   a. **Select PERSPECTIVE (3D) from View menu. A 3D view will appear on the screen with a wind direction arrow and a ground plane north arrow.**

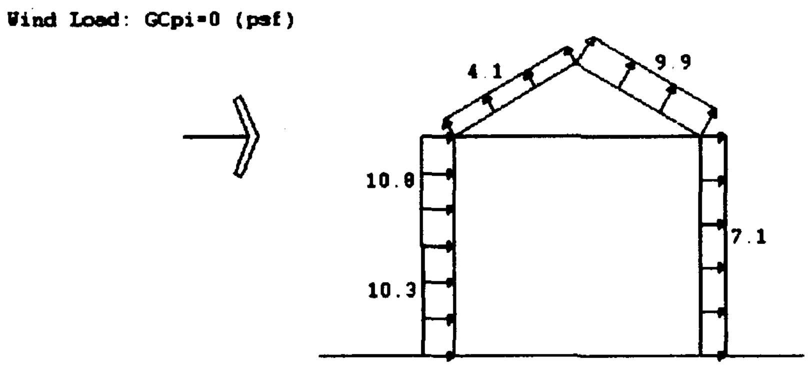


» **Note: Pressures are shown in a cyan (light blue) color, and suctions are shown in a magenta (purple) color.**



   b. **Rotate the view direction 90 degrees clockwise by dragging the black arrow around the circle in the View Direction tool.**

   c. **Select SECTION from the Options pull-down menu. The rotated 2-D plan and section will appear with new wind pressures and suctions shown.**



» **Note: The wind direction is always shown left to right in the section view.**

2. **Display wind loads with consideration of internal pressures.**

   a. **Select SHOW LOADS from the View menu. A Show Loads dialog window will appear.**

   b. **Select either GCpi Negative or GCpi Positive .**

   c. **Click on OK . The selected wind loads will appear on the section.**

   d. **Repeat steps a through c to view the other GCpi case.**

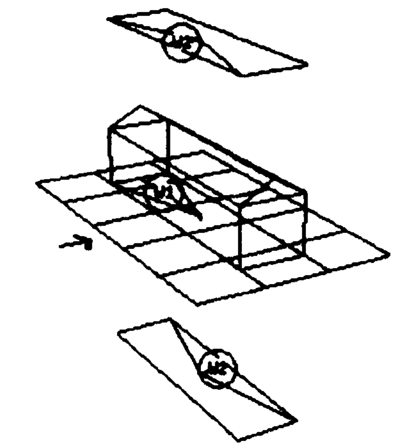
3. **Review B and L Assumptions used in wind calculations.**

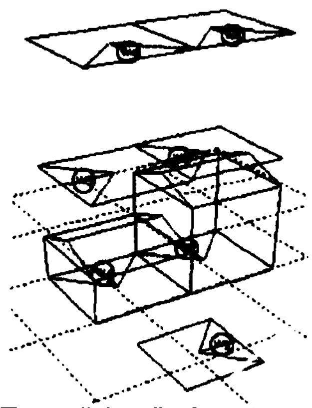
   a. **Select PERSPECTIVE (3D) .**

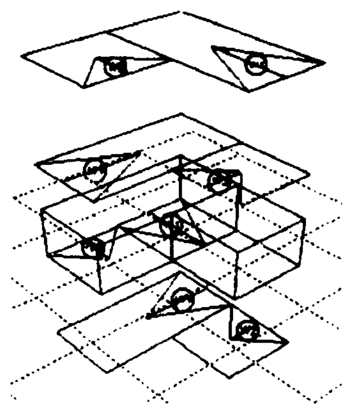
   b. **Select B & L ASSUMPTIONS from the Show Loads window.**

c. Click on OK. Red B & L rectangles for wind in all four directions will appear.

» **Note: For Example 1 refer to the B and L diagram shown on the right. Each rectangular outline represents the assumed B and L dimensions used in the calculations. The wind direction is shown by a triangle pointing in the direction of the wind. The numbered labels appearing on each red rectangle coincide with the numbered titles on the output calculations. To display all four rectangles, you will need to zoom out. Refer to the B&L examples below for irregular shapes.**







4. Turn off the display of openings.

   a. Select SHOW STRUCTURE from the View pull-down menu. A Show Structure pop-up dialog window will appear.
   b. Turn off OPENINGS.
   c. Select OK and the displayed openings on the 3-D model will disappear.

5. To rotate, pan, zoom, change from wireframe to solid, and perform other operations with the 3-D model; follow steps D-1 through D-6 of Example 1 in the snow load section.

E. Generation of hard copies.

1. For details on the following commands, refer to steps E-1 through E-4 of Snow Load Example 1.
   a. View → Section
   b. File → Print Screen
      ✔ Printer
      OK
   c. File → Print Data
      ✔ Wind GCpi

✔ **Wind**

✔ **Print to File**

✔ **Execute Notepad**

OK

d. **Notepad → File → Page Setup**

Left Margin : 0.5

Right Margin : 0

OK

e. **Notepad → File → Print**

f. **Notepad → File → Exit**

Computed Percentage of Openings output:

```
Project     : Industrial Building
Location    : Huntsville
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991  2:39 PM

****************** Internal Pressure Coefficients, GCpi ******************
                          Condition                                  GCpi
---------------------------------------------------------------------------
1. Percentage of openings in one wall exceeds that of all other
   walls by 10 percent or more, and openings in all other walls
   do not exceed 20 percent of respective wall area.
   Percentage of openings is based on gross area of wall.    +0.75 & -0.25

2. All other cases.                                          +0.25 & -0.25
---------------------------------------------------------------------------

                                            Wall     Opening  Percentage of
Wall Plane Name                           Area (sf) Area (sf)  Opening (%)
---------------------------------------------------------------------------
Wall - 1                                    420.0      40.0       9.52
Wall - 2                                   1350.0     150.0      11.11
Wall - 3                                    420.0      40.0       9.52
Wall - 4                                   1350.0     650.0      48.15
---------------------------------------------------------------------------

Wall - 4 satisfies condition 1.
+-------------------------+
| GCpi = +0.75 and -0.25  |
+-------------------------+
```

» **Note: Wind output titles include a number which corresponds to the number on the B & L Assumptions rectangles. This will assist the user in interpreting the direction of wind in the calculations.**

Main Force Resistance System Output:

```
Project     : Industrial        g
Location    : Huntsville
Design Load : TM 5-809-1 198b
Time        : Thu Sep 12, 1991  2:39 PM

***************************** Wind Load - 1 *****************************

Velocity Importance Exposure  Width     Length                Roof Type
           Factor            Perpend.  Parallel
                             to Wind   to Wind
  (mph)                        (ft)      (ft)
---------------------------------------------------------------------------
  70.0     1.00        C      20.0      75.0

Distance to ocean line >= 100 mi.   h/d = 1.05 <= 5
```

************************* Main Framing Pressures *************************

Parallel to Ridge or Length

Location	z or h (ft)	Gh	Kz	qz (psf)	Cp	External GCpi=0	Pressure -0.25	P (psf) 0.75
Windward Wall								
level 3	24.0	1.29	0.92	11.5	0.80	11.9	14.6	3.6
level 2 - 3	21.0	1.29	0.88	11.0	0.80	11.4	14.1	3.1
level 1 - 2	9.0	1.29	0.80	10.0	0.80	10.3	13.1	2.1
level 1	0.0	1.29	0.80	10.0	0.80	10.3	13.1	2.1
Leeward Wall	21.0	1.29	0.88	11.0	-0.21	-3.0	-0.2	-11.2
Side Wall	21.0	1.29	0.88	11.0	-0.70	-9.9	-7.2	-18.2
Roof	21.0	1.29	0.88	11.0	-0.70	-9.9	-7.2	-18.2
Internal	21.0		0.88	11.0		0.0	-2.8	8.3

***************************** Wind Load - 2 *****************************

Velocity (mph)	Importance Factor	Exposure	Width Perpend. to Wind (ft)	Length Parallel to Wind (ft)	Roof Type
70.0	1.00	C	75.0	20.0	Gable

WW : 7.20 in 12
Distance to ocean line >= 100 mi. h/d = 1.05 <= 5 Lee: 7.20 in 12

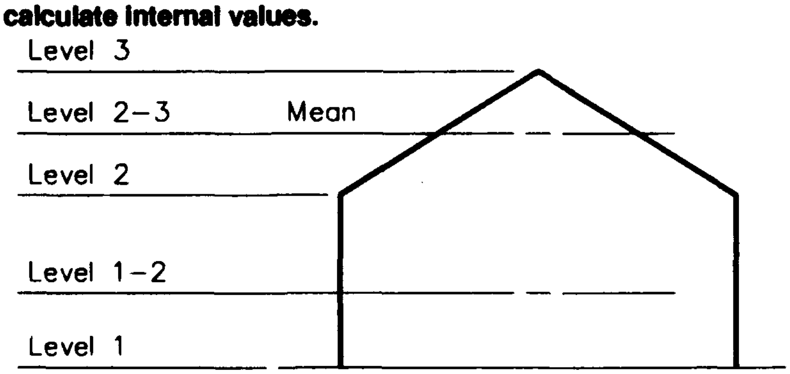
************************* Main Framing Pressures *************************

Perpendicular to Ridge or Length

Location	z or h (ft)	Gh	Kz	qz (psf)	Cp	External GCpi=0	Pressure -0.25	P (psf) 0.75
Windward Wall								
level 2	18.0	1.29	0.84	10.5	0.80	10.8	13.6	2.6
level 1 - 2	9.0	1.29	0.80	10.0	0.80	10.3	13.1	2.1
level 1	0.0	1.29	0.80	10.0	0.80	10.3	13.1	2.1
Leeward Wall	21.0	1.29	0.88	11.0	-0.50	-7.1	-4.3	-15.3
Side Wall	21.0	1.29	0.88	11.0	-0.70	-9.9	-7.2	-18.2
Windward Roof	21.0	1.29	0.88	11.0	-0.29	-4.1	-1.4	-12.4
Leeward Roof	21.0	1.29	0.88	11.0	-0.70	-9.9	-7.2	-18.2
Roof Parallel	21.0	1.29	0.88	11.0	-0.70	-9.9	-7.2	-18.2
Internal	21.0		0.88	11.0		0.0	-2.8	8.3

Notes for main framing:
Positive pressures act toward surfaces.
Pressure or suction = P = q*Gh*Cp and/or P = qh*Gh*Cp-qh*(GCpi)

» **Note: Levels on the windward side are designated by numbers starting with 1 at elevation z = 0.0 feet. Level 2 becomes the eave height at elevation z = 18.0 feet, since no intermediate floors were created in the model. Level 3 is the ridge height for wind parallel to the ridge, which is z = 24.0 feet. Level 1-2 indicates a midheight level between levels 1 and 2. This corresponds to elevation z = 9.0 feet in this example. Level 2-3 also indicates a midheight level between levels 2 and 3, which in this case is also the mean roof elevation of 21.0 feet. The mean roof height is used to calculate pressures and suctions for leeward, side, and roof surfaces. It is also used to calculate internal values.**

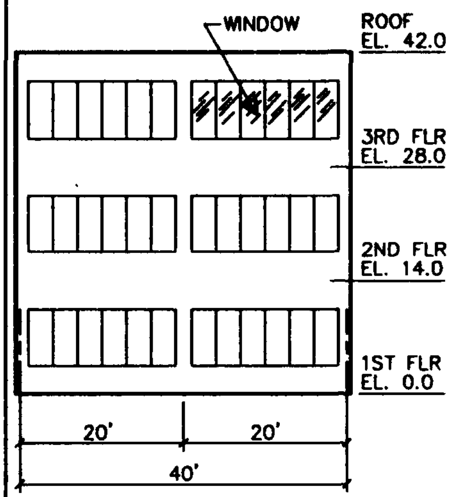


F. Save the building model with its wind loads applied for future reference.

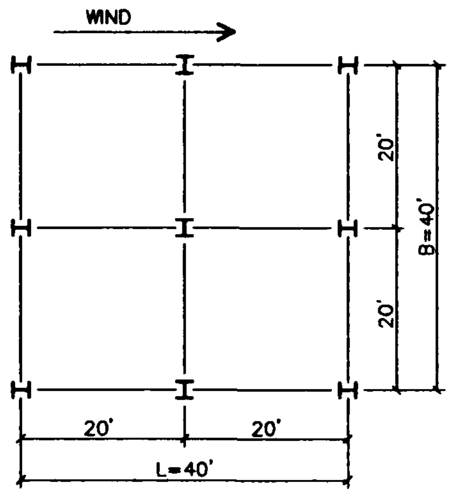
1. **Refer to steps F-1 through F-4 in Snow Load Example 1 for details on the following commands.**
   a. File → Save
      Enter file name
      OK

EXAMPLE TWO: THREE STORY FLAT ROOF

Given: This example is taken from page D-14 of TM 5-809-1 1986. It is a three-story administrative building with a height less than 60 feet. It is sited in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico. It is assumed to be in exposure category C and to have an importance factor of I. The windows shown on the elevation exist on all four elevations. A plan and elevation follow:



ELEVATION



FLOOR PLAN

Required: Determine the external wind pressure and suctions on all surfaces.

Solution:

An abbreviated discussion is given here since most of the steps repeat in a similar fashion to Example One.

A. Establish Criteria

1. Input the following data into the PROJECT, REGIONAL, and SITE CRITERIA dialog windows:

PROJECT	Project name	: Administration Bldg.
	City/Installation	: Ammo Plant
	State	: MS
	Design Load	: TM5-809-1 1986
REGIONAL	Basic Wind Speed	: 100.0 mph
	Coastal	: yes (highlight with an X)
SITE WIND	Importance	: Category I
	Exposure	: Category C
	Distance to Oceanline	: 0 miles

B. Draw volumetric model.

1. **Select the DRAW MODEL tool palette.**

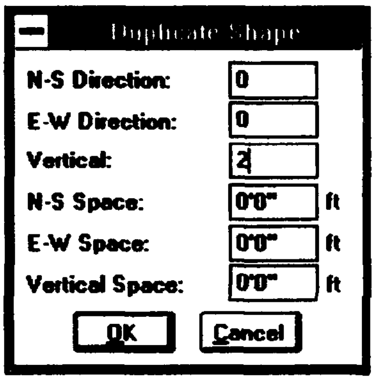
2. **Create the first floor building volume.**
   a. **Establish general layout requirements as required.**

» **Hint: start with the default initial shape size and drag planes or set the initial object size to the correct proportions.**

   b. **Add the Cube to the ground plane.**

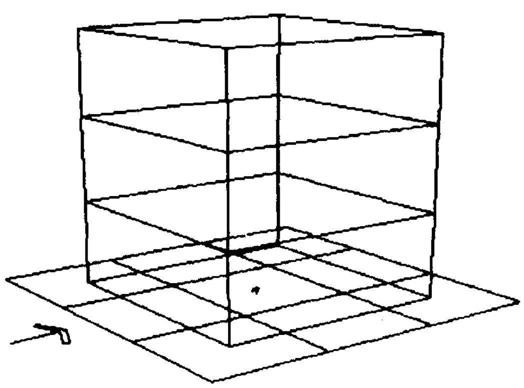
3. **Create the second and third floor volumes.**
   a. **Select DUPLICATE SHAPE from the Edit pull-down menu or click on the Rabbits icon. A Duplicate pop-up dialog window will appear asking you how many objects are to be added in each direction and the clear spacing between the objects.**

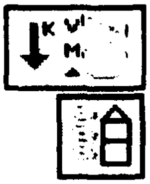
   b. **Change the VERTICAL number to 2. Leave the VERTICAL SPACE distance at 0.0 feet so that the objects will connect directly.**



   c. **Click on OK and solid square handles will appear on the visible planes of the shape.**
   d. **Select any one of the three handles on the shape and the two vertically duplicated shapes will appear. Handles will now appear on all three shapes to permit duplicating another shape.**
   e. **Double click on the right mouse key to exit the Duplicate command.**

4. **This completes the model for this example.**



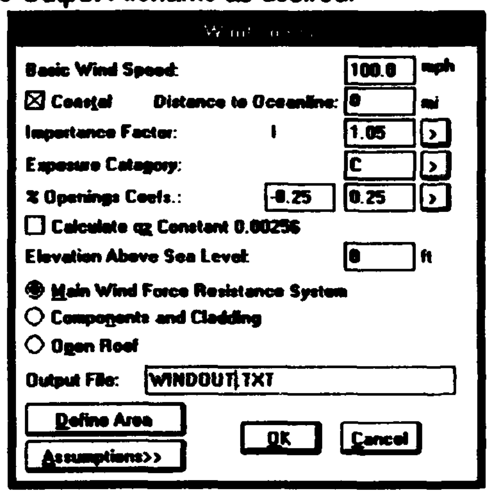


C. Develop Main Wind Force Resistance wind loads on the building.

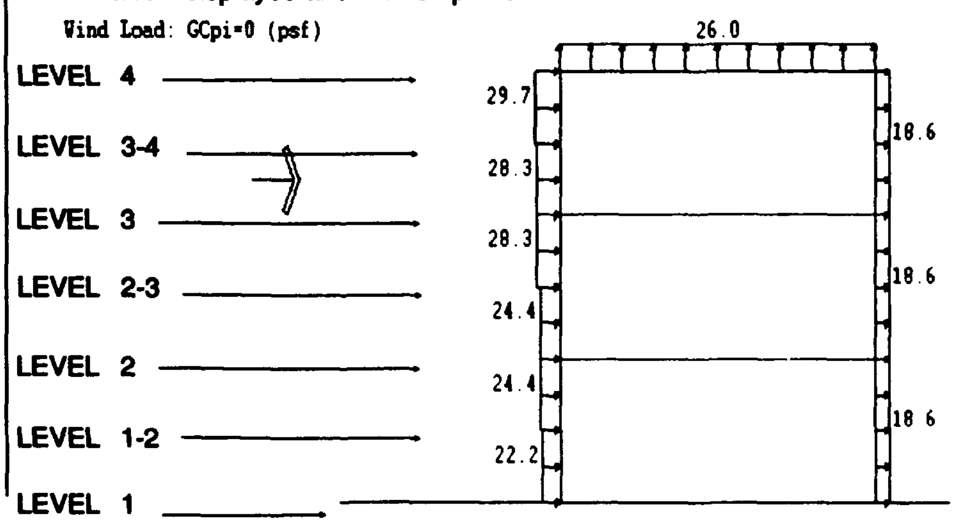
1. Select the LOADS AND DESIGN tool palette.
2. Select WIND from the Loads pull-down menu or the Wind icon. The Wind Loads pop-up dialog window will appear.
   a. The % OPENINGS COEFS. are set at the default values of +0.25 and -0.25.

» **Note: It is not necessary to draw in the openings on the building and have the computer check code criteria and select coefficients when it is obvious that similar openings exist on all four elevations.**

   b. Turn off CALCULATE QZ CONSTANT 0.00256.
   c. Turn on MAIN WIND FORCE RESISTANCE SYSTEM.
   d. Modify the Output Filename as desired.



3. Click on OK when satisfied, and wind load calculations will begin. A pop-up dialog window will keep you informed of the progress.
4. The building plan and a section will appear upon completion of the calculations. Wind pressures and suctions will be shown for the wind direction arrow displayed and with GCpi = 0.



D. Manipulation of the building model and its wind loads.

1. **Refer to steps D-1 through D-5 from Example 1 in this section for details on the following commands.**
   a. **View → Perspective 3D**
   b. **Rotate view 90 degrees**
   c. **View → Section**
   d. **View → Show Loads**
      ✔ **GCpi Negative**
      OK
   e. **View → Show Loads**
      ✔ **GCpi Positive**
      OK
   f. **View → Perspective 3D**
   g. **View → Show Loads**
      ✔ **B&L Assumptions**
      OK
   h. **View → ShowStructure**
      Turn off openings
      OK
   i. **Rotate 3D view**
   j. **Pan 3D view**
   k. **Zoom 3D view**
   l. **View → Solid Object**
   m. **Perform any other operations with the 3D model**

E. Generate hard copies.

Wind Load Example 2 sample output:

```
Project      : Administration Bldg.
Location     : Ammo Plant
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   3:23 PM

****************************** Wind Load - 1 ******************************

Velocity  Importance  Exposure   Width     Length               Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                          (ft)      (ft)
-----------------------------------------------------------------------------
  100.0      1.05         C      40.0      40.0

Distance to ocean line:   0 mi.    h/d = 1.05 <= 5

************************ Main Framing Pressures *************************

Parallel to Ridge or Length
-----------------------------------------------------------------------------
Location            z or h   Gh    Kz     qz     Cp    External Pressure P (psf)
                     (ft)                (psf)          GCpi=0   -0.25    0.25
-----------------------------------------------------------------------------
Windward Wall
  level  4           42.0   1.23  1.07   30.2   0.80    29.7     37.3     22.2
  level  3 - 4       35.0   1.23  1.02   28.8   0.80    28.3     35.9     20.8
  level  2 - 3       21.0   1.23  0.88   24.8   0.80    24.4     32.0     16.9
  level  1 - 2        7.0   1.23  0.80   22.6   0.80    22.2     29.8     14.7
  level  1            0.0   1.23  0.80   22.6   0.80    22.2     29.8     14.7
Leeward Wall         42.0   1.23  1.07   30.2  -0.50   -18.6    -11.0    -26.1
Side Wall            42.0   1.23  1.07   30.2  -0.70   -26.0    -18.5    -33.6
```

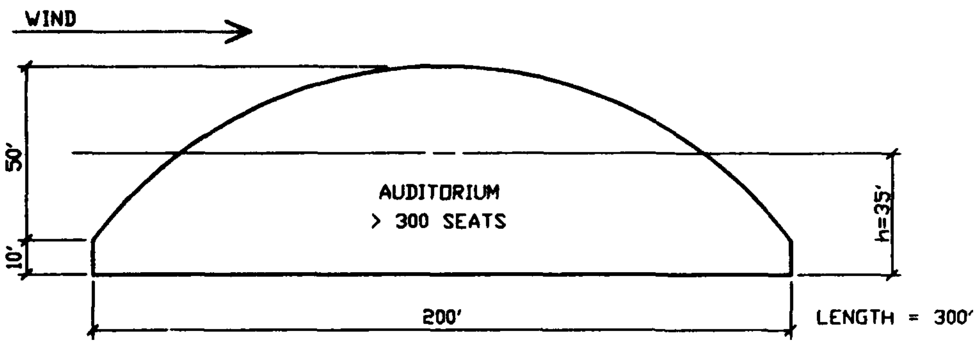
```
Roof                 42.0  1.23  1.07  30.2 -0.70    -26.0     -18.5     -33.6
Internal             42.0        1.07  30.2            0.0      -7.6       7.6

Notes for main framing:
   Positive pressures act toward surfaces.
   Pressure or suction = P = q*Gh*Cp and/or P = qh*Gh*Cp-qh*(GCpi)
```

>> External wind pressure and suction values are typically computed for wind applied parallel and perpendicular to the long dimension or ridge of a building. The answers are the same in this example since it is a square in plan.

F. Save the building model with its wind loads applied for future reference.

[illegible]

Given: This example is taken from page D-18 of TM 5-809-1 1986. It is a one-story auditorium with a barrel vault roof. It is sited at Robbins AFB, GA, and has an assumed exposure category of C. It has a seating capacity of more than 300 people, which requires an importance category II. The percentage of openings is no more than 10%. A typical elevation follows:



Required: Determine the external wind pressures and suctions for all surfaces for wind applied parallel and perpendicular to the crown.

Solution:

An abbreviated discussion is given here since most of the steps repeat in a similar fashion to Example 1.

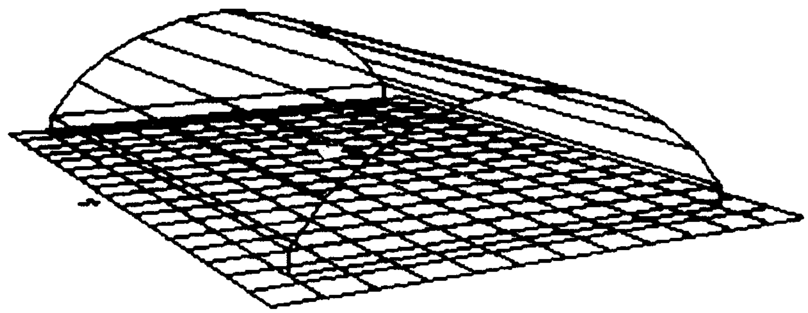
A. Establish Criteria

1. Input the following data into the PROJECT, REGIONAL, and SITE pop-up dialog windows from Criteria on the CASM menu bar.

PROJECT	Project Name	:Auditorium
	City/Installation	:Robbins AFB
	State	:GA
	Design Load	:TM5-809-1 1986
REGIONAL	Basic Wind Speed	:75 mph
	Coastal	:no (leave blank)
SITE WIND	Importance	:Category II
	Exposure	:Category C
	Distance to Coastline	:100 miles (default)

B. Draw volumetric model

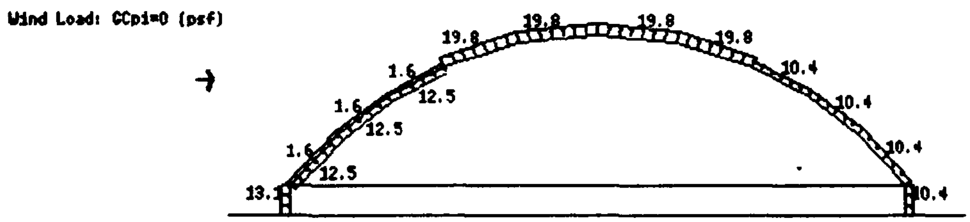
1. Follow the procedure outlined in step B of Example 2 from the snow load section, or any appropriate combination of commands from previous examples to create the basic geometry required.



C. Develop Main Wind Force Resistance wind loads on the building.

D. Manipulate the building model and its wind loads.

E. Generate hard copies.



» **The output includes wind pressures and suctions on all surfaces for the cases of wind applied perpendicular and parallel to the crown. Values for the arch are broken into windward quarter (two values), middle half, and leeward quarter. Three columns of values are provided to account for no consideration of internal pressure ($GC_{pi} = 0$), and positive and negative internal pressures considered ($GC_{pi} = -0.25$ or $+0.25$). The internal pressure coefficients were selected by you. A sample output follows:**

Wind Load Example 3 sample output:

```
Project      : Auditorium
Location     : Robbins AFB
Design Load  : TM 5-809-1 1986
Time         : Thu Sep 12, 1991   3:34 PM

****************************** Wind Load - 1 ******************************

Velocity  Importance  Exposure   Width     Length             Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                          (ft)      (ft)
---------------------------------------------------------------------------
   75.0     1.07         C       200.0     300.0

Distance to ocean line >= 100 mi.   h/d = 0.18 <= 5
```

************************* Main Framing Pressures *************************

Parallel to Ridge or Length

Location	z or h (ft)	Gh	Kz	qz (psf)	Cp	External GCpi=0	Pressure P (psf) -0.25	0.25
Windward Wall								
level 3	60.0	1.24	1.19	19.6	0.80	19.4	23.6	15.2
level 2 - 3	35.0	1.24	1.02	16.8	0.80	16.7	20.9	12.5
level 1 - 2	5.0	1.24	0.80	13.2	0.80	13.1	17.3	8.9
level 1	0.0	1.24	0.80	13.2	0.80	13.1	17.3	8.9
Leeward Wall	35.0	1.24	1.02	16.8	-0.40	-8.3	-4.1	-12.5
Side Wall	35.0	1.24	1.02	16.8	-0.70	-14.6	-10.4	-18.8
Roof	35.0	1.24	1.02	16.8	-0.70	-14.6	-10.4	-18.8
Internal	35.0		1.02	16.8		0.0	-4.2	4.2

****************************** Wind Load - 2 ******************************

Velocity (mph)	Importance Factor	Exposure	Width Perpend. to Wind (ft)	Length Parallel to Wind (ft)	Roof Type
75.0	1.07	C	300.0	200.0	Arched

Crown: 50.0 ft
Rise-to-Span ratio r = 0.25   0.2 <= r < 0.3
Distance to ocean line >= 100 mi.   h/d = 0.18 <= 5

************************* Main Framing Pressures *************************

Perpendicular to Ridge or Length

Location	z or h (ft)	Gh	Kz	qz (psf)	Cp	External GCpi=0	Pressure P (psf) -0.25	0.25
Windward Wall								
level 2	10.0	1.24	0.80	13.2	0.80	13.1	17.3	8.9
level 1	0.0	1.24	0.80	13.2	0.80	13.1	17.3	8.9
Leeward Wall	35.0	1.24	1.02	16.8	-0.50	-10.4	-6.2	-14.6
Side Wall	35.0	1.24	1.02	16.8	-0.70	-14.6	-10.4	-18.8
Windward Quarter	35.0	1.24	1.02	16.8	0.08	1.6	5.8	-2.6
Windward Quarter *	35.0	1.24	1.02	16.8	-0.60	-12.5	-8.3	-16.7
Middle Half	35.0	1.24	1.02	16.8	-0.95	-19.8	-15.6	-24.0
Leeward Quarter	35.0	1.24	1.02	16.8	-0.50	-10.4	-6.2	-14.6
Internal	35.0		1.02	16.8		0.0	-4.2	4.2

Note:
* Windward quarter roof pressure has 2 values per Table 5-5.

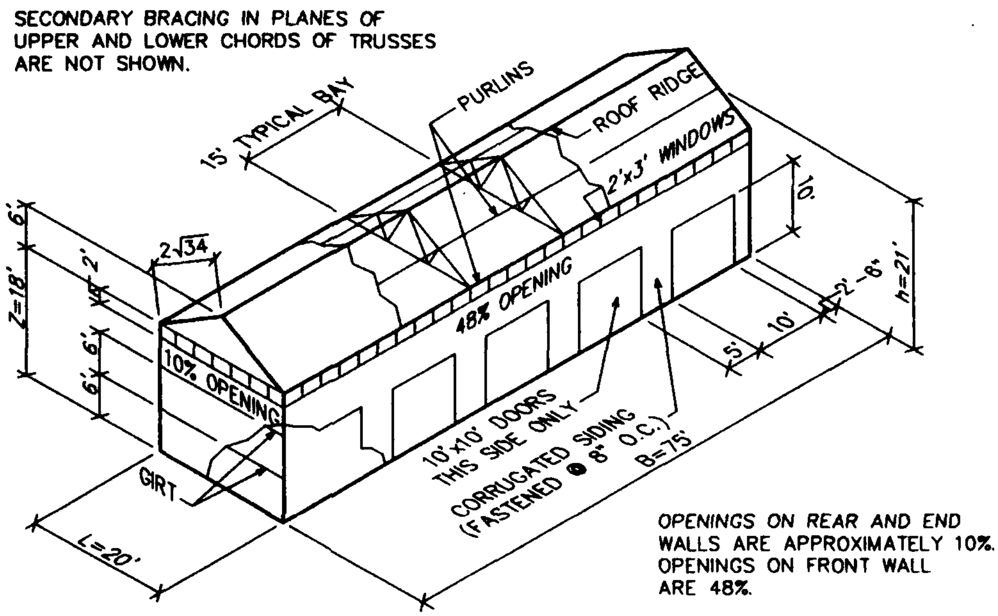
Notes for main framing:
Positive pressures act toward surfaces.
Pressure or suction = P = q*Gh*Cp and/or P = qh*Gh*Cp-qh*(GCpi)

F. Save the building model with its wind loads applied for future reference.

■ Components and Cladding

EXAMPLE ONE: Building height less than 60 feet

Given: This example uses all the building data of Example One for Main Wind-Force Resisting Systems and continues the example on page D-1 of TM 5-809-1 1986.



Required: Determine the pressures and suctions on the following building components:

a. Overhead door adjacent to building corner.

b. Typical corner wall girt.

c. Maximum tension on a wall fastener.

Solution:

A. Establish Criteria.

1. Retrieve the saved file of main wind force resisting system (Example 1).
   a. Select OPEN from the File pull-down menu on the CASM menu bar. A pop-up dialog window will appear with a list of all saved files.
   b. Scroll if necessary to find the desired filename.
   c. Use the mouse to highlight the filename and select OK to load the data file.
2. Review the PROJECT, REGIONAL, and SITE Criteria windows to ensure that the desired values are present.

B. Draw volumetric model.

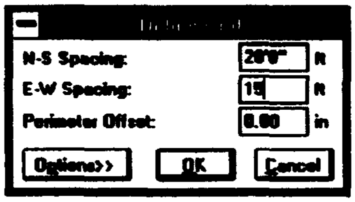
1. The model is already complete for this example and appears on the screen.
2. Select PERSPECTIVE 3D.
3. Turn off the display of main-force resisting loads with the SHOW LOADS command.
4. Turn on the display of openings with the SHOW STRUCTURE command.

C. Draw structural girts as shown on the given isometric drawing.

1. Select the DRAW STRUCTURE tool palette.
2. Develop a structural grid within which structural elements can be drawn.

» It is necessary to establish a structural grid before structural elements can be inserted.

a. Select DEFINE GRID from the Grid/Opening pull-down menu. A Define Grid dialog window will appear with default values.



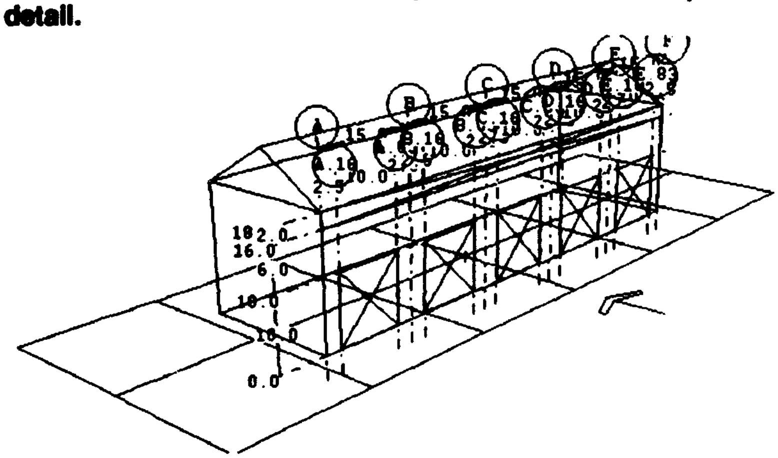
b. Set N-S SPACING to 20 feet and the E-W SPACING to 15 feet which is the typical bay for this example.

c. Leave the PERIMETER OFFSET at 0.0 inch to indicate that the grid coincides with the extreme outer surface of the exterior envelope.

d. Select OK to define the grid on the building volume. The grid will appear on the currently selected plane.

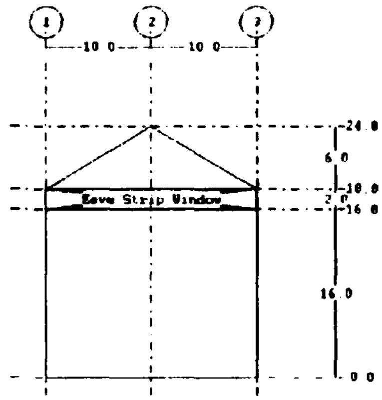
» The grid will appear on the last vertical plane upon which work was done .

» Note: The Grid OPTIONS selection is not needed for this example since we will use the default lettering, numbering, and bubble locations. The Reference manual explains the available options in more detail.



3. **Select the end wall vertical plane which is to receive the girts.**
   a. **Select <u>VERTICAL STRUCTURAL PLANE</u> from the View pull-down menu. Handles will appear on the visible vertical planes.**
   b. **Select the end wall vertical plane with the mouse.**

The plane will appear in 2-D with its openings and pertinent grids with their dimensions.



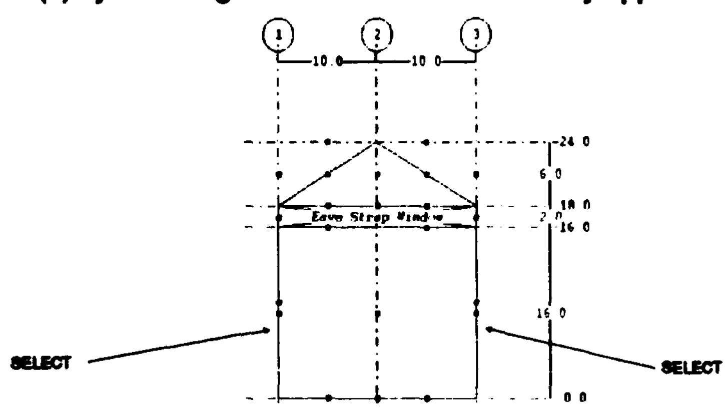
4. **Draw the two widely spaced horizontal girts on the end wall.**
   a. **Select <u>WIDELY SPACED</u> from the Surface/Linear pull-down menu. Handles appear at the midpoints of the gridline segments and the midpoints of the modeled plane lines.**

» **Note: Surface/Linear contains surface and linear structural elements which can be placed within horizontal, inclined, and vertical planes.**

   b. **Select a sufficient number of handles in a <u>clockwise order</u> to define the perimeter within which structure is to be drawn. The perimeter should not include the opening. A highlighted dotted line will be drawn showing the perimeter.**

» **Note: A handle represents the midpoint of a line which contains two points on one edge of the perimeter. The perimeter is made up of these points.**

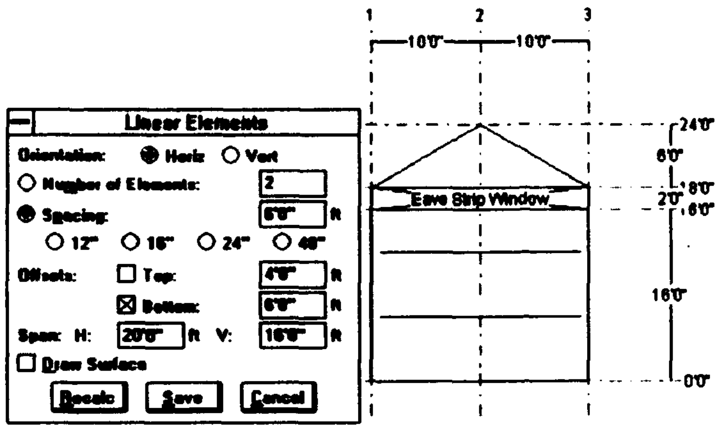
» **Note: There are several ways to select the desired perimeter: (1) by selecting handles in a clockwise order around the entire perimeter, or (2) by selecting one handle and the mutually opposite side handle.**



» **Note: If you select an incorrect handle, double click the right mouse key to stop adding to the perimeter and select CANCEL in the element dialog box.**

c. **When the desired perimeter is selected, double click the right mouse key to fix the perimeter. The Linear Elements dialog window appears and a single widely spaced linear element appears on the screen.**

» **Note: The spans indicated in the dialog window are calculated from the selected perimeter.**



d. **Revise data in the dialog window to draw two girts spaced 6 feet apart and 6 feet from grade as follows:**

(1) **Set ORIENTATION to HORIZ.**

(2) **Fix the SPACING and set to 6.0 feet.**

» **Note: Linear elements are placed by fixing the spacing or fixing the number of elements.**

» **Note: A checkmark in front of a variable fixes that variable.**

(3) **Fix the BOTTOM OFFSET and set to 6.0 feet.**

» **Note: If neither Offset is fixed, the elements are centered within the perimeter.**

(4) **Turn off DRAW SURFACE.**

(5) **Click on RECALC to redraw the girts at the new settings. The Number of Elements will be calculated as 2 and the Top Offset as 4.0 feet.**

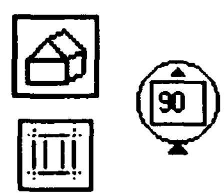
(6) **Click on SAVE to fix the two girts.**

5. **Draw two girts on the long elevation without the door openings.**

a. **Return to the PERSPECTIVE (3D) view of the model.**

b. **Rotate the 3-D view to make the backside elevation visible.**

c. **Follow steps 3 and 4 above to insert two horizontal girts 15 feet long and spaced as the girts on the end elevation. Place the girts in the far right end bay.**



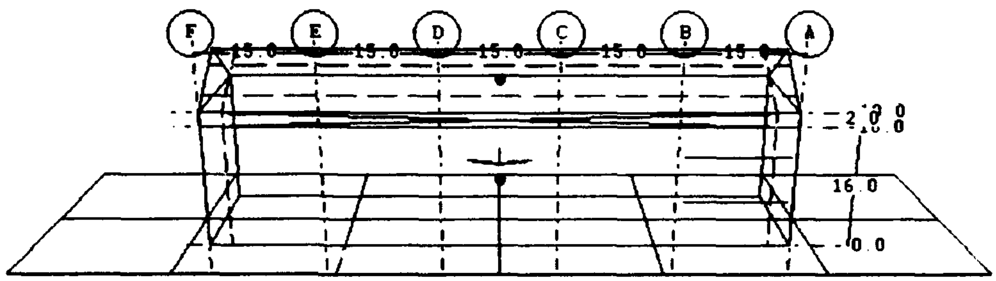


D. Develop Components and Cladding wind loads on the 15-foot-long girt.

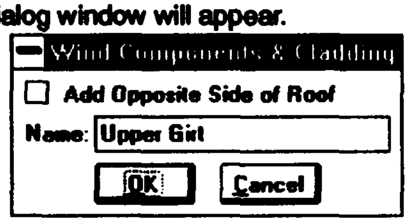
1. Switch to a PERSPECTIVE (3D) view of the model.

>> **Note: It is necessary to be in the 3-D mode to calculate components and cladding wind loads.**

2. Select the LOADS AND DESIGN tool palette.
3. Select WIND from the Loads pull-down menu. A Wind Loads pop-up dialog window will appear.
   a. Verify the wind load criteria.
   b. Turn on COMPONENTS AND CLADDING.
   c. Revise OUTPUT FILE name as: GIRT.TXT.
   d. Click on OK to begin calculation of the "a" edge distances. A dialog window will appear to keep you informed of the progress. When finished, the 3-D model will display the "a" distances by dashed red lines. Handles will appear on the visible surfaces.

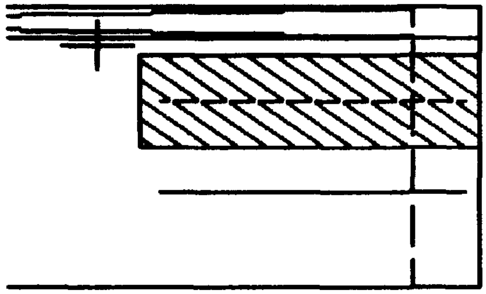


4. Select one of the handles on the plane which contains the two 15-foot-long girts. A 2-D elevation of the plane and a Tributary Area dialog window will appear.
5. Create the tributary area for the upper girt.
   a. Move the mouse pointer to a distance from the lower left corner of 9.0 feet vertically and 75.0 feet horizontally.
   b. Click the left mouse key to fix the lower right corner of the tributary area.
   c. Move the mouse pointer to dynamically expand the rectangle to 15 feet horizontally and 6 feet vertically.
   d. Click the left mouse key to fix the tributary area. A Wind Components and Cladding dialog window will appear.



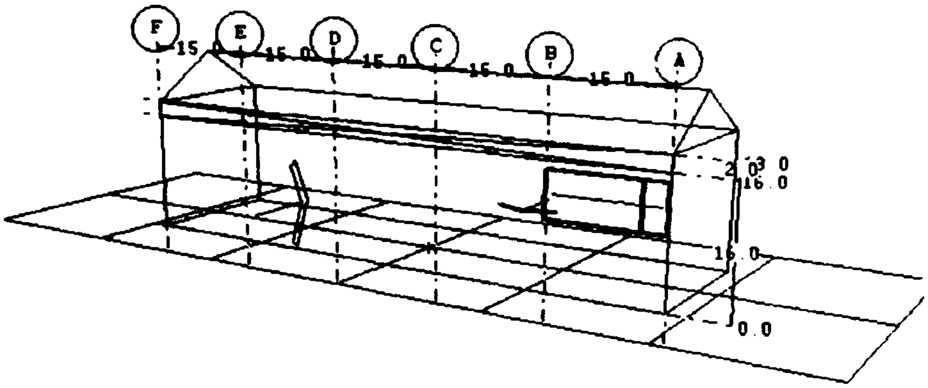
   e. Enter the name "Upper Girt", turn OFF ADD OPPOSITE SIDE OF ROOF, and click on OK. A blue hatched rectangle will denote the tributary area.

>> **Note: Selection of CANCEL will not add the tributary area.**



6. Double click the right mouse key to end creation of tributary areas. Wind load calculations are now performed for the component. A 3-D view of the wind load on the girt will appear when calculations are completed.

» **Note: The wind direction is set perpendicular to the plane that contains the component and is pointed to create a windward load.**



E. Develop Components and Cladding wind loads for one fastener on the end elevation with the two girts.

1. Select WIND from the Loads pull-down menu.
2. Verify the information shown in the Wind Loads dialog window and change the OUTPUT FILE name to: FASTENER.TXT.
3. Click on OK when satisfied and "a" distance calculations begin. When completed, the "a" distances and handles will appear on the 3-D model.
4. Select one of the end elevation handles and a 2-D elevation will appear as will a Tributary Area dialog window.
5. Place a fastener tributary area with dimensions of 8 inches (0.67 foot) horizontally and 6 feet vertically with its lower left corner a distance of 4 inches (0.33 foot) from the left edge of the elevation and 3 feet up from grade.

» **Note: The actual tributary length is 8 inches, yet the prescribed minimum length is 2 feet for tributary area calculations on fasteners.**

6. Select CANCEL from the Wind Components and Cladding dialog window to not add this incorrect tributary area. The drawn tributary area will be erased from the model.

7. Redo step 5 for a tributary area with dimensions 2 feet horizontally and 6 feet vertically using the same lower left corner location.
8. Name the component: FASTENER in the Wind Components and Cladding dialog window and select OK to add this desired tributary area. The area will become hatched on the 2-D elevation.
9. Double click the right mouse key to stop adding tributary areas. Wind load calculations are now performed for the component. A 3-D view of the wind load for the fastener will appear when calculations are completed.

» **Note: The graphical depiction of the wind load on the girt is no longer shown since for this wind direction the girt is on a side wall. Side wall pressures and suctions are not included in code requirements for consideration of wind on components and cladding.**

F. **Develop Components and Cladding wind loads for the door adjacent to the end elevation with the two girts.**

1. Rotate the model to reveal the long elevation with the overhead doors.

» **Note: The doors on that elevation will not appear on the 3-D model since it was not the last structural plane selected. The openings will appear when the component and cladding plane is selected.**

2. Select WIND from the Loads pull-down menu.
3. Verify the information in the Wind Loads dialog window and change the OUTPUT FILE name to: DOOR.TXT.
4. Click on OK when satisfied and "a" distance calculations begin. When completed, the "a" distances and handles will appear on the 3-D model.
5. Select the elevation handle and a 2-D elevation will appear as will a Tributary Area dialog window.
6. Place a door tributary area with dimensions of 10 feet horizontally and 10 feet vertically with its lower left corner a distance of 2.5 feet from the left edge of the elevation and 0.0 foot vertically (at grade).
7. Name the component: DOOR in the Wind Components and Cladding dialog window and select OK to add this desired tributary area. The area will become hatched on the 2-D elevation.
8. Double click the right mouse key to stop adding tributary areas. Wind load calculations are now performed for the component. A 3-D view of the wind load for the door will appear when calculations are completed.

» **Note: Because of the new wind direction, the girt's tributary area wind suction is displayed on the leeward elevation. The fastener's tributary area wind load is not displayed since it is now on a side wall which is parallel to the wind direction.**

G. **This completes drawing and calculation for component and cladding wind loads on all three components.**

H. **Manipulation of the building model and all of its wind loads.**

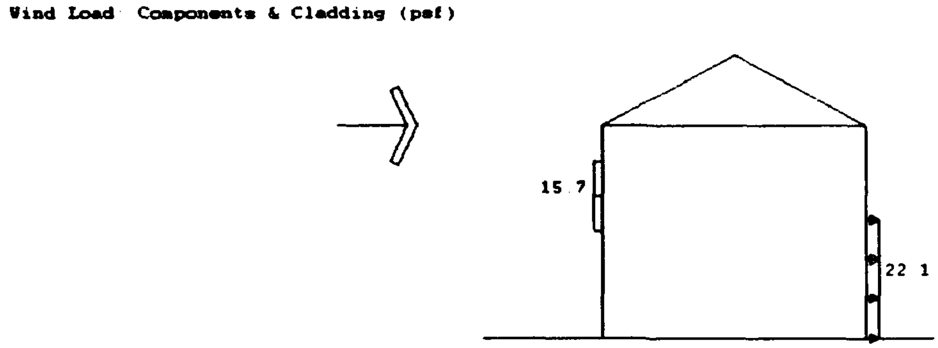
1. Take a section cut to view the girt and door components and cladding wind values.

» **Note: Presently, only vertical section cuts can be taken.**

a. **Select various cursor positions to view wind magnitudes inside and outside of the "a" distance.**

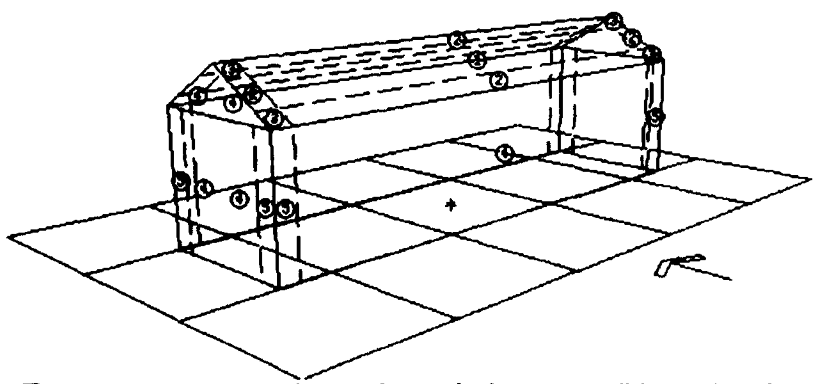
b. **Return to the 3-D model and rotate the model 180 degrees.**

c. **Take another section cut to view the wind magnitudes on the girt and door, when the wind comes from the opposite direction.**



2. **Repeat step 1 for the fastener wind loads.**
3. **View the code-prescribed zonal areas for components and cladding.**
   a. **Return to a 3-D view of the model.**
   b. **Select SHOW LOADS from the Viewpoint Options pull-down menu. A Show Loads dialog window will appear.**
   c. **Turn on ZONE AREAS.**
   d. **Click on OK and the zone areas with their circled number will appear on the 3-D model. Red dashed lines separate the zone areas.**

» **Note: The numbers shown on the model and used in the output calculations correspond to the specific zones established in TM 5-809-1.**



4. **To rotate, pan, zoom, change from wireframe to solids, and perform other operations with the 3-D model; follow steps D-1 through D-4 of Example 1 in the snow loads section.**

I. **Generate hard copies.**

J. **Save the building model with its wind loads applied for future use.**

GIRT.TXT Output File:

```
Project     : Industrial Building
Location    : Huntsville
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991   4:13 PM

********************************* Wind Load *********************************

Velocity  Importance  Exposure   Width     Length                 Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                           (ft)      (ft)
-----------------------------------------------------------------------------
   70.0     1.00          C       75.0      20.0
Distance to ocean line >= 100 mi.   h/d = 1.05 <= 5

 Height      Kh       qh          GCpi
  (ft)               (psf)
-------------------------------------------------
  21.0      0.88     11.0     -0.25  0.75

Height <= 60 ft

****************** Component/Cladding Pressures (psf) *******************

           -----------------------------Walls-----------------------------
                   Windward                                Leeward
Tributary     Zone 4            Zone 5            Zone 4            Zone 5
Area (sf)     middles           corners           middles           corners
           GCp      P        GCp      P        GCp      P        GCp      P
           -------------------------------------------------------------------
Internal          -2.8              -2.8                8.3               8.3
Upper Girt
  90.0     1.18   15.7      1.18   15.7     -1.28  -22.3     -1.49  -24.6
      a = 3.0 ft

Notes for components and cladding:
   P = qh(GCp)-qh(GCpi)
   Internal pressures have been included in above values.
   * for roof overhangs: algebraically add this pressure
     to the above values.  P = qh(GCp) = 0.8qh
```

FASTENER.TXT Output File:

```
Project     : Industrial Building
Location    : Huntsville
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991   4:22 PM

********************************* Wind Load *********************************

Velocity  Importance  Exposure   Width     Length                 Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                           (ft)      (ft)
-----------------------------------------------------    -------------------
   70.0     1.00          C       20.0      75.0

Distance to ocean line >= 100 mi.   h/d = 1.05 <= 5

 Height      Kh       qh          GCpi
  (ft)               (psf)
-------------------------------------------------
  21.0      0.88     11.0     -0.25  0.75

Height <= 60 ft

****************** Component/Cladding Pressures (psf) *******************

           -----------------------------Walls-----------------------------
                   Windward                                Leeward
Tributary     Zone 4            Zone 5            Zone 4            Zone 5
Area (sf)     middles           corners           middles           corners
           GCp      P        GCp      P        GCp      P        GCp      P
           -------------------------------------------------------------------
Internal          -2.8              -2.8                8.3               8.3
Fastener
  12.0     1.38   17.9      1.38   17.9     -1.48  -24.5     -1.96  -29.8
      a = 3.0 ft

Notes for components and cladding:
   P = qh(GCp)-qh(GCpi)
   Internal pressures have been included in above values.
   * for roof overhangs: algebraically add this pressure
     to the above values.  P = qh(GCp) = 0.8qh
   To comply with TM 5-809-1, wall external pressures
   have not been reduced 10% per ANSI figure 3, note 3.
```

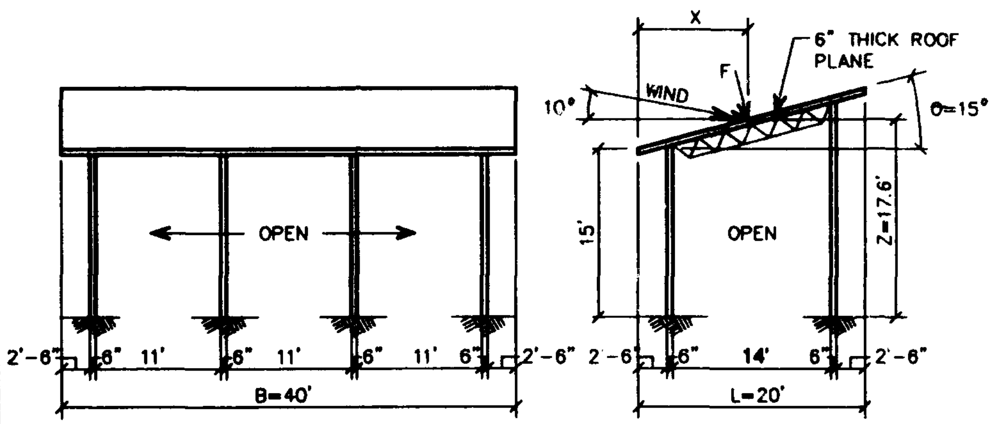
DOOR.TXT Output File:

```
Project     : Industrial Building
Location    : Huntsville
Design Load : TM 5-809-1 1986
Time        : Thu Sep 12, 1991  4:26 PM

******************************** Wind Load ********************************

Velocity  Importance  Exposure   Width     Length              Roof Type
           Factor              Perpend.  Parallel
                               to Wind   to Wind
 (mph)                          (ft)      (ft)
---------------------------------------------------------------------------
  70.0      1.00        C       75.0      20.0

Distance to ocean line >= 100 mi.   h/d = 1.05 <= 5

 Height     Kh       qh        GCpi
  (ft)              (psf)
--------------------------------------------
  21.0     0.88     11.0    -0.25  0.75

Height <= 60 ft

****************** Component/Cladding Pressures (psf) *******************

           ----------------------------Walls-----------------------------
                     Windward                           Leeward
Tributary    Zone 4           Zone 5            Zone 4           Zone 5
Area (sf)    middles          corners           middles          corners
            GCp     P        GCp     P         GCp     P        GCp     P
           ---------------------------------------------------------------
Internal          -2.8              -2.8                8.3              8.3
Door
 100.0      1.16   15.5     1.16   15.5      -1.26  -22.1     -1.47  -24.4
      a = 3.0 ft

Notes for components and cladding:
   P = qh(GCp)-qh(GCpi)
   Internal pressures have been included in above values.
   * for roof overhangs: algebraically add this pressure
     to the above values.  P = qh(GCp) = 0.8qh
```

■ Unenclosed Buildings

EXAMPLE ONE: One-story, Monoslope roof

Given: This one-story open-sided structure example is taken from page D-20 of TM 5-809-1 1986. It is an open storage facility located at Hickman AFB, Honolulu, HI. The wind exposure category is D, and the importance category is I. An elevation and section are shown below:



Required: Determine the design roof wind force, F.

Solution:

An abbreviated discussion of input is given here since many of the steps repeat in a similar fashion to Example 1 in the Main Wind Force Resisting System section.

A. Establish Criteria

1. Input the following data into the PROJECT, REGIONAL, and SITE CRITERIA dialog windows:

PROJECT	Project Name	:	Open Storage
	City/Installation	:	Hickman AFB
	State	:	HI
	Design Load	:	TM 5-809-1 1986
REGIONAL	Basic Wind Speed	:	80 mph
	Coastal	:	no (leave blank)
SITE WIND	Importance	:	Category I
	Exposure	:	Category D
	Distance to Oceanline	:	100 miles (default)

B. Draw volumetric model

» **Note: The drawing of open structures involves the use of planes and columns rather than the cube and prism shapes used to create solid (enclosed) buildings.**

1. Select the <u>DRAW MODEL</u> tool palette.

» **Note: There are many ways to construct this open structure; however, it is recommended to begin with the plane and add the columns as a second step.**

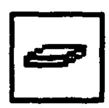
2. Establish general layout requirements which are different than previously established.

   a. Use the following:

DEFINE UNITS(snap increment)			: 6 inches
SNAP TO UNITS			: on
SHOW GROUND PLANE			: on
GROUND PLANE			
	WIDTH	N-S	: 100 feet
		E-W	: 100 feet
	SPACING	N-S	: 20 feet
		E-W	: 20 feet
INITIAL SHAPE SIZE			
	N-S WIDTH		: 20 feet
	E-W WIDTH		: 40 feet
	HEIGHT		: 25 feet
	PLANE THICKNESS		: 6 inches
	COLUMN WIDTH		: 6 inches
	ORIENTATION		: E-W
STACK ON GROUND PLANE			: on
DIRECTIONS LOCKED			: none

» **Note: The Initial Object Size Plane Thickness becomes the horizontal plane thickness and the E-W Width becomes the length of the roof. The Initial Object Size Height for the column is arbitrary, but it must be high enough to extend the columns through the horizontal (roof) plane.**

3. Select <u>HORIZONTAL PLANE</u> from the Shapes pull-down menu. A horizontal plane will appear on the ground plane. A Dimensions dialog window will also appear.



   a. Hold down the right mouse key while dragging the mouse toward and away from you to elevate the plane a TRANSLATED DISTANCE vertically of 14.5 feet. Keep the N-S and E-W Translated Distances at 0.0 foot.

» **Note: The N-S and E-W directions could have been locked under the Layout discussion to prevent the plane from translating in these two directions while elevating the plane.**

» **Note: The 14.5-foot vertical dimension is to the underside of the plane.**

b. Click the left mouse key to fix the plane.

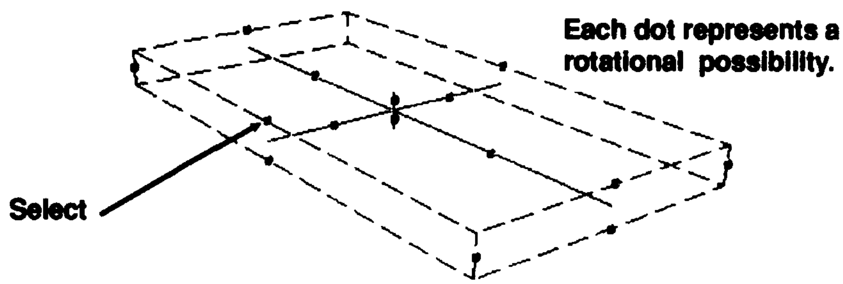
c. Double click the right mouse key to stop adding planes.

4. Rotate the plane 15 degrees.

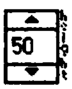
a. Select ROTATE SHAPE from the Edit pull-down menu. Handles appear at the centroid of all visible planes on the object.

b. Select one of the handles with the mouse pointer. The shape becomes highlighted and handles appear on the edges and on the centroidal axes of the shape about which the desired rotation can be selected.

Oops! Handles on the top and bottom edges overlap due to the view being quite distant.



c. Double click the right mouse key to cancel the Rotate command.



d. Zoom the plane toward you with by selection of the HEIGHT and DISTANCE tools. Strive for an approximate height of 30 feet and a distance of 70 feet at a view direction of 40 degrees.

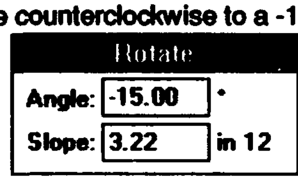
» Note: The view height, distance, and direction values can be typed in.

» Note: Slowly zoom and rotate incrementally when using the single-screen option of CASM due to the slow graphic response of the VGA system.

e. Select ROTATE SHAPE and one of the handles on the plane again. The shape becomes highlighted, and this time all the handles are visible.

f. Select the upper handle of the left side long dimension of the plane with the mouse pointer. All the handles will disappear, and a Rotate dialog window will appear.

g. Move the mouse left-right to rotate the plane about the selected edge. Rotate the plane counterclockwise to a -15.0 degree angle.



h. Click the left mouse key to fix the position of the plane. Handles will again appear for additional rotation operations.

i. Double click the right mouse key to exit the rotation command.

j. Adjust the view back in space to see the whole plane.

5. Insert a temporary cube to create the roof plane projected width of 20 feet.

a. Select CUBE from the Shapes pull-down menu. The shape appears on the screen at its initial Shape Size previously set in Layout.

b. Do not Translate the object. Translated Distance values in the Dimensions dialog window should remain at 0.0 foot.

c. Click the left mouse key to fix the cube.

» **Note: The cube will seem to disappear because the next shape falls directly on top of it.**

d. Double click the right mouse key to exit the command. The cube will reappear.

6. Increase the width of the roof plane to facilitate slicing with the cube.

a. Select DRAG PLANE from the Edit pull-down menu. Handles will appear on the visible planes to drag.

» **Note: Each Plane shape is composed of six planes, four of which are its edges and two of which are surfaces.**

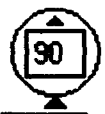
b. Use the mouse pointer to select the handle which corresponds to the edge plane of the roof plane. The plane will be highlighted and the Dimensions dialog window will appear.

c. Move the mouse toward and away from you to Drag the plane beyond the extremity of the cube.

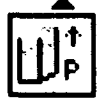
d. Click the left mouse key to fix the plane's position.

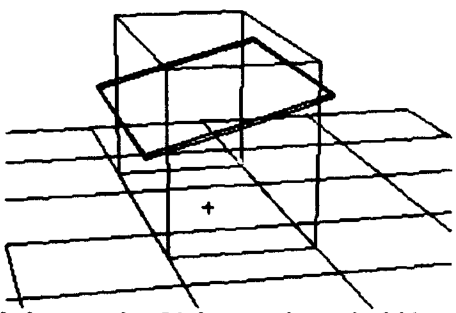
e. Double click the right mouse key to exit the command.

f. Rotate the view of the model to make the opposite side of the roof visible by use of the View Direction tool.



g. Redo steps a through e to extend the roof plane beyond the extremity of the cube.





7. Slice the roof plane to the 20 foot-projected width.

a. Select SLICE SHAPE from the Edit pull-down menu. Handles will appear on the visible planes of each shape.

b. Select the shape to be sliced. Use the mouse pointer to select one of the roof plane handles. The shape will become highlighted.

c. Select the plane to do the slicing. Use the mouse pointer to select the long vertical plane on the cube. The selected plane is highlighted and the shape is sliced into two parts. New handles appear for another object to be sliced.

d. Double click the right mouse key to exit the Slice command.

e. Rotate the model to view the opposite side of the roof plane by use of the View Direction tool.

f. Redo steps a through d to slice the roof plane on this side.

8. Delete the unwanted parts of the sliced roof plane.

a. Select DELETE SHAPE from the Edit pull-down menu. Handles will appear on the visible planes of the shapes and the sliced parts.

b. Use the mouse pointer to select one of the handles on one of the unwanted sliced parts. The part will be deleted.

c. Select the other unwanted sliced part of the roof plane.

d. Select a handle on the cube since it is no longer needed.

e. Double click the right mouse key to exit the Delete command.

9. Insert the first column in the geometric model.

» **Note: The columns are not required for the wind analysis of an open structure in the CASM program. They are drawn here for graphical completeness and to illustrate the required column modeling commands.**

a. Insert the column shape.

(1) Select COLUMN from the Shapes pull-down menu. A column appears on the ground plane to the proportions set in layout, and a Dimensions dialog window will appear.

(2) Position the first column somewhere in the southwest corner of the roof plane.

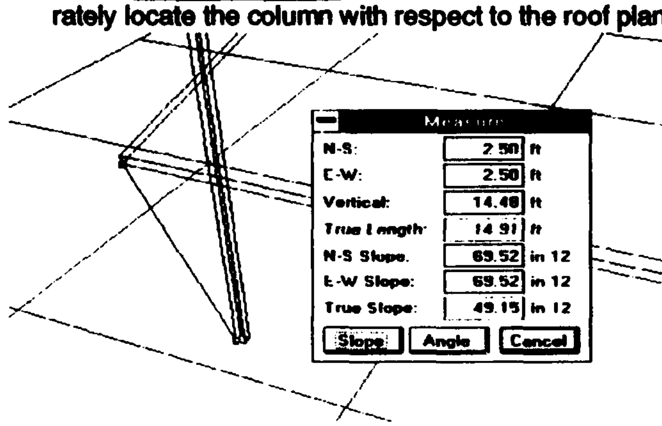
(3) Click the left mouse key to insert the column.

(4) Double click the right mouse key to exit the Column command.

b. Select the vertices between which measurements are to be taken to locate the position of the column.

» **Note: You may wish to zoom in closer to the plane and column.**

(1) Select TAPE MEASURE from the Edit pull-down menu to accurately locate the column with respect to the roof plane.



(2) Select a southwest corner vertex of the roof plane with the mouse pointer. A red dot will appear to highlight the vertex.

» **Note: The vertex selected is the one closest to the point of the mouse pointer or the center of the cross hairs.**

(3) Select the southwest corner vertex of the column (top or bottom) by clicking the left mouse key when satisfied. The selected vertex is highlighted, a dotted red line will connect the two selected vertices, and a Measure dialog window will appear. The values therein represent the relationship between the two vertices.

» **Note: There are four vertices at each end of the column, not just one.**

» **Note: Switch to the 2-D Plan view and Zoom window to verify that the correct column vertex has been selected and then switch back to the 3-D view.**

c. Select DEFINE UNITS and set the SNAP INCREMENT to 3 inches.

d. Move the column to its correct location.

(1) Select MOVE SHAPE from the Edit pull-down menu and handles will appear on the visible planes of the shapes.

(2) Use the mouse pointer to select a handle on the column. The column will be highlighted.

(3) Move the mouse to drag the column to a position N-S of 2.5 feet and E-W of 2.5 feet.

» **Note: Watch the dynamic change in values within the Measure dialog window, rather than be fooled by the position of the column in the perspective view.**

(4) Click the left mouse key to fix the column location.

(5) Double click the right mouse key to exit the Move Shape command.

e. Select CANCEL from the Measure dialog window to stop measuring between the two vertices.

10. Duplicate the first inserted column at the northwest corner of the roof.

a. Select DUPLICATE SHAPE from the Edit pull-down menu. A Duplicate Shape dialog window will appear.

b. Enter the following data:

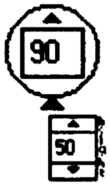
N-S DIRECTION	: 1
E-W DIRECTION	: 0
VERTICAL	: 0
N-S SPACE	: 14.0 feet
E-W SPACE	: 0.0 feet
VERTICAL SPACE	: 0.0 feet

» **Note: The Space values represent clear distances, not centerline distances.**

c. Click on OK and handles will appear on the visible surfaces of the shapes.

d. Select one handle on the column. The shape will be duplicated one time 14 feet away.

e. Double click the right mouse key to exit the Duplicate Shape command.



11. Slice the two columns with the bottom surface of the roof plane.

a. Rotate the view of the model to a worm's eye perspective looking up at the underside of the roof plane.

» **Note: It is possible to verify the wireframe view by switching to the solid view.**

b. Select SLICE SHAPE from the Edit pull-down menu. Handles will appear on the visible surfaces of the shapes.

c. Select the column shape to be sliced with the mouse pointer. The column will be highlighted.

d. Select the bottom surface of the roof plane to slice the column with the mouse pointer. The plane is highlighted and the column is sliced into two parts.

e. Select the other column to be sliced. It will be highlighted.

f. Select the bottom surface of the roof plane. The plane will be highlighted and the column will be sliced into two parts.

g. Double click the right mouse key to exit the Slice Shape command.

12. Use DELETE SHAPE to remove the two unwanted upper parts of the two columns.

13. Duplicate the remaining six columns.

a. Select DUPLICATE SHAPE from the Edit pull-down menu. A Duplicate dialog window will appear.

b. Enter the following data:

N-S DIRECTION	: 0
E-W DIRECTION	: 3
VERTICAL	: 0
N-S SPACE	: 0.0 feet
E-W SPACE	: 11.0 feet
VERTICAL SPACE	: 0.0 feet

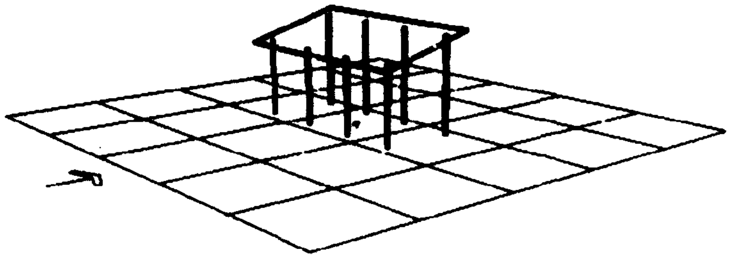
c. Click on OK and handles will appear on the visible surfaces of the shapes.

d. Select a handle on one of the columns. The shape will be duplicated three times and spaced 11 feet apart.

e. Select the handle on the other column and it will be duplicated three times and spaced 11 feet apart.

f. Double click the right mouse key to exit the Duplicate Shape command.

14. This completes creation of the model.



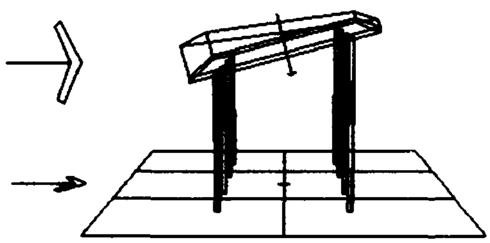
C. **Develop the open structure wind forces on the shed roof.**

1. Select the LOADS AND DESIGN tool palette.

» **Note: A 3-D view must appear on the screen to be able to calculate open roof wind loads.**

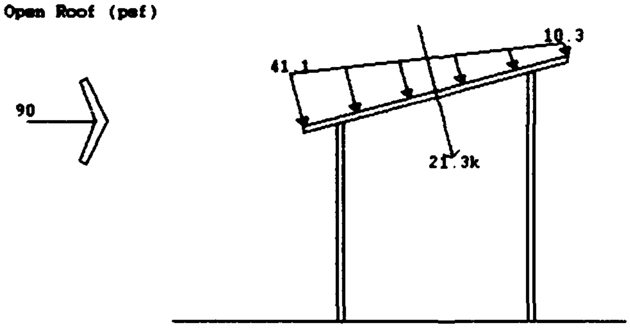
» **Note: You must position the 3-D view of the model to see the top surface of the roof in order to activate the wind load calculations.**

2. Select WIND from the Loads pull-down menu. A Wind Loads dialog window will appear.
3. Verify values in the Wind Loads dialog window and turn on OPEN ROOF. Modify any values as desired.
4. Click on OK for CASM to begin finding the Open Roof planes. Handles will appear on the open planes found.
5. Use the mouse pointer to select the roof plane to receive wind loads. Wind calculations are then performed on the open plane. A 3-D depiction of the wind load will appear on the model when calculations are completed.

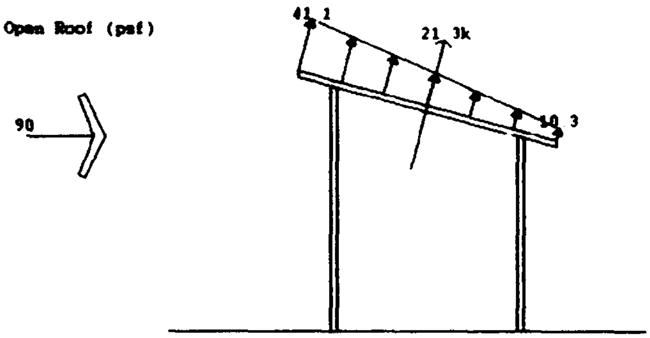


D. **Manipulate the building model and its wind loads.**

Wind Load: Open Roof (psf)



Wind Load: Open Roof (psf)



E. Generate hard copies.

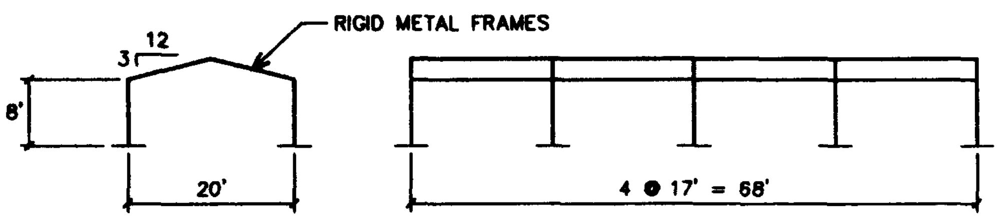
Unenclosed Buildings Example 1 sample output:

```
Project      : Open Storage
Location     : Hickman AFB
Design Load  : TM 5-809-1 1986
Time         : Fri Sep 13, 1991   9:16 AM

********************************* Wind Load *********************************

Velocity  Importance  Exposure   Width     Length               Roof Type
            Factor               Perpend.  Parallel
                                 to Wind   to Wind
  (mph)                           (ft)      (ft)
-----------------------------------------------------------------------------
  80.0      1.00         D        40.0      20.0              Flat/Monoslope
                                                                3.22 in 12
Distance to ocean line >= 100 mi.

************************ Open Roof Pressures (psf) ************************

z = h = 17.68 ft
Gh = 1.15
Kz = 1.24
qz = 0.00256*Kz*(I*V)*(I*V) = 20.30 psf
Af = [L/cos(theta)]*B = 828.2 sqft
B/L = 2.00
Cf = 1.10
X/L = 0.40
theta+10 deg = 25.0 deg
                     Pressure on top of roof     Pressure on bottom of roof
-----------------------------------------------------------------------------
X                     8.00 ft from low eave        8.00 ft from high eave
F = qz*Gh*Cf*Af               21.30 k                     -21.30 k
-----------------------------------------------------------------------------
P1 (leeward edge)   = [2*F*cos(theta)/(B*L)]*[3*X/L-1] =  10.30 psf
P2 (windward edge)  = [2*F*cos(theta)/(B*L)]*[2-3*X/L] =  41.10 psf

Notes for open roof pressures:
   Positive pressures act toward surfaces.
```

F. Save the building model with its wind loads applied for future reference.

EXAMPLE: Wind on an Open-Gabled Roof

Given: The one story open-gabled roof carport shown below. It is located at the Chanute AFB in Rantoul, IL. The importance category is I and the exposure category is C.



Required: Determine the design wind pressures on the roof.

Solution:

The minimum recommended exterior force coefficients for such an open gabled roof are extracted from NAVFAC DM 2.2, STRUCTURAL ENGINEERING LOADS, DESIGN MANUAL 2.2, NOVEMBER 1981, and referenced in TM 5-809-1 1986 on page 5-12. These recommended coefficients are not included in ANSI A58.1-1982.

A. Establish Criteria.

1. Input the following data into the PROJECT, REGIONAL, and SITE Criteria dialog windows:

PROJECT:	Project Name	: Carport
	City/Installation	: Chanute AFB - Rantoul
	State	: IL
	Design Load	: TM 5-809-1 1986
REGIONAL:	Basic Wind Speed	: 70 mph
	Coastal	: No
SITE WIND:	Importance	: Category I
	Exposure	: Category C
	Distance to Oceanline	: 100 mile

B. Draw volumetric model

1. Select the DRAW MODEL tool palette.

2. Establish general layout requirements which are different than previously established.

   a. Use the following:

DEFINE UNITS(snap increment)			: 3 inches
SNAP TO UNITS			: on
SHOW GROUND PLANE			: on
GROUND PLANE			
	WIDTH	N-S	: 100 feet
		E-W	: 100 feet
	SPACING	N-S	: 20 feet
		E-W	: 20 feet
INITIAL SHAPE SIZE			
	N-S WIDTH		: 10 feet
	E-W WIDTH		: 68 feet
	HEIGHT		: 20 feet
	PLANE THICKNESS		: 6 inches
	COLUMN WIDTH		: 6 inches
	ORIENTATION		: E-W
STACK ON GROUND PLANE			: on
DIRECTIONS LOCKED			: none

3. Select HORIZONTAL PLANE from the Shapes pull-down menu. A horizontal plane will appear on the ground plane. A Dimensions dialog window will also appear.
   a. Hold down the right mouse key while dragging the mouse toward and away from you to elevate the plane a TRANSLATED DISTANCE vertically of 7.5 feet, N-S distance of -5.0 feet, and an E-W distance of 0.0 feet.

» **Note: The vertical dimension is to the underside of the plane.**

   b. Click the left mouse key to fix the plane. A second plane will appear on the ground plane ready for positioning next to the first plane.
   c. Hold down the right mouse key while dragging the mouse toward and away from you to elevate the second plane a TRANSLATED DISTANCE vertically of 7.5 feet, N-S distance of 5.0 feet, and an E-W distance of 0.0 foot.
   d. Click the left mouse key to fix the plane.
   e. Double click the right mouse key to stop adding planes.
4. Drag the common edges of the two planes to create the ridge for the roof with a slope of 3 in 12.

» **Note: It is necessary to drag the top and bottom edge of each plane; thus, four edges will be elevated to create the ridge of the roof.**

   a. LOCK the N-S and E-W directions.
   b. Hide the plane closest to you.

» **Note: This enables viewing one plane's edges at a time.**

      (1) Select HIDE SHAPES from the View pull-down menu located in the Viewpoint window. Handles will appear on the visible planes of each shape.

(2) Select one of the handles on the shape closest to you and the shape will be hidden. Handles will appear on the remaining shapes to permit more shapes to be hidden.

(3) Double click on the right mouse key to exit the Hide Shapes command.

C. Activate the Tape measure command.

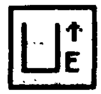
1. Select TAPE MEASURE from the Edit pull-down menu.

2. Use the mouse pointer to select the two top vertices along the short edge of the plane. Red dots will appear at the two vertices and a Measure dialog window will also appear.

D. Drag the two common edges of the horizontal plane.

1. Zoom in on the view of the plane to space the edges farther apart so each edge handle will be visible.

2. Select DRAG EDGE from the Edit pull-down menu. Handles appear on the visible edges to drag.



>> Note: If both the top and bottom edge handles do not appear, it will be necessary to cancel the command and repeat step 1 to further separate the edges.

3. Select the top edge with the mouse pointer. The edge will be highlighted.

4. Hold down the right mouse key while dragging the mouse toward and away from you to elevate the edge to a roof slope of 3 in 12. The slope is dynamically adjusted in the Measure window as the mouse is moved. The translated vertical distance is shown in the Dimensions and Measure windows as 2.5 feet.

5. Click the left mouse button to fix the edge.

6. Select the bottom edge of the plane and drag it to a translated vertical distance of 2.5 feet as displayed in the Dimensions window.

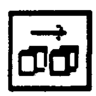
>> Note: None of the values in the Measure window will change, since they apply to the already elevated edge.

7. Click the left mouse key to fix the bottom edge.

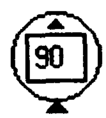
8. Double click the right mouse key to exit the Drag Edge command.

9. Select CANCEL on the Measure window to stop measuring between the two vertices.

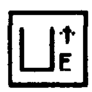
10. Select SHOW SHAPES from the View pull-down menu to have the hidden objects reappear.



11. Rotate the view of the model so the plane containing the remaining ridge edges is visible.



12. Use the DRAG EDGE command to elevate these edges 2.5 feet vertically (a slope of 3 in 12).



13. UNLOCK the N-S and E-W directions.

E. Insert the first column in the geometric model.

» **Note: The columns are not required for the wind analysis of an open structure in the CASM program. They are drawn here for graphical completeness and to illustrate the required column modeling commands.**

1. Insert the first column shape.

» **You may wish to rotate the view so you are looking north.**

a. Select COLUMN from the Shapes pull-down menu. A column appears on the ground plane to the proportions set in layout, and a Dimensions dialog window will appear.

b. Position the first column somewhere in the southwest corner of the gable roof.

c. Click the left mouse key to insert the column.

d. Double click the right mouse key to exit the Column command.

2. Select the vertices between which measurements are to be taken to locate the position of the column.

a. Select TAPE MEASURE from the Edit pull-down menu to accurately locate the column with respect to the roof plane.

b. Select a southwest corner vertex of the roof plane with the mouse pointer. A red dot will appear to highlight the vertex.

» **Note: The vertex selected is the one closest to the point of the mouse pointer or the center of the cross hairs.**

c. Select the southwest corner vertex of the column by clicking the left mouse key when satisfied. The selected vertex is highlighted, a dotted red line will connect the two selected vertices, and a Measure dialog window will appear. The values therein represent the relationship between the two vertices.

» **Note: There are four vertices at each end of the column, not just one.**

» **Note: Switch to the 2-D plan view to verify that the correct column vertex has been selected, and then switch back to the 3-D view.**

3. Move the column to its correct location.

a. Select MOVE SHAPE from the Edit pull-down menu and handles will appear on the visible planes of the shapes.

b. Use the mouse pointer to select a handle on the column. The column will be highlighted.

c. Move the mouse to drag the column to a position N-S of 0.0 foot and E-W of 0.0 foot.

» **Note: Watch the dynamic change in values within the Measure dialog window, rather than be fooled by the position of the column in the perspective view.**

d. Click the left mouse key to fix the column location.

e. Double click the right mouse key to exit the Move Shape command.

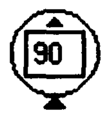
4. Select CANCEL from the Measure dialog window to stop measuring between the two vertices.
5. Duplicate the first inserted column at the northwest corner of the roof.

   a. Select DUPLICATE SHAPE from the Edit pull-down menu. A Duplicate Shape dialog window will appear.
   b. Enter the following data:

N-S DIRECTION	: 1
E-W DIRECTION	: 0
VERTICAL	: 0
N-S SPACE	: 19.0 feet
E-W SPACE	: 0.0 feet
VERTICAL SPACE	: 0.0 feet

» Note: The Space values represent clear distances, not centerline distances.

   c. Click on OK and handles will appear on the visible surfaces of the shapes.
   d. Select one handle on the column. The shape will be dupicated one time 19 feet away.
   e. Double click the right mouse key to exit the Duplicate Shape command.
6. Slice the two columns with the bottom surfaces of the roof planes.



   a. Rotate the view of the model to a worm's eye perspective looking up at the underside of both roof planes.

» Note: It is possible to verify the wireframe view by switching to the solid view.

   b. Select SLICE SHAPE from the Edit pull-down menu. Handles will appear on the visible surfaces of the shapes.
   c. Select the column shape to be sliced with the mouse pointer. The column will be highlighted.
   d. Select the bottom surface of the roof plane that intersects the column to slice the column with the mouse pointer. The plane is highlighted and the column is sliced into two parts.
   e. Select the other column to be sliced. It will be highlighted.
   f. Select the bottom surface of the other roof plane. The plane will be highlighted, and the column will be sliced into two parts.
   g. Double click the right mouse key to exit the Slice Shape command.

7. Use DELETE SHAPE to remove the two unwanted upper parts of the two columns.
8. Duplicate the remaining eight columns.

   a. Select DUPLICATE SHAPES from the Edit pull-down menu. A Duplicate Shapes dialog window will appear.
   b. Enter the following data:

N-S DIRECTION	: 0
E-W DIRECTION	: 4
VERTICAL	: 0
N-S SPACE	: 0.0 feet
E-W SPACE	: 16.375 feet
VERTICAL SPACE	: 0.0 feet

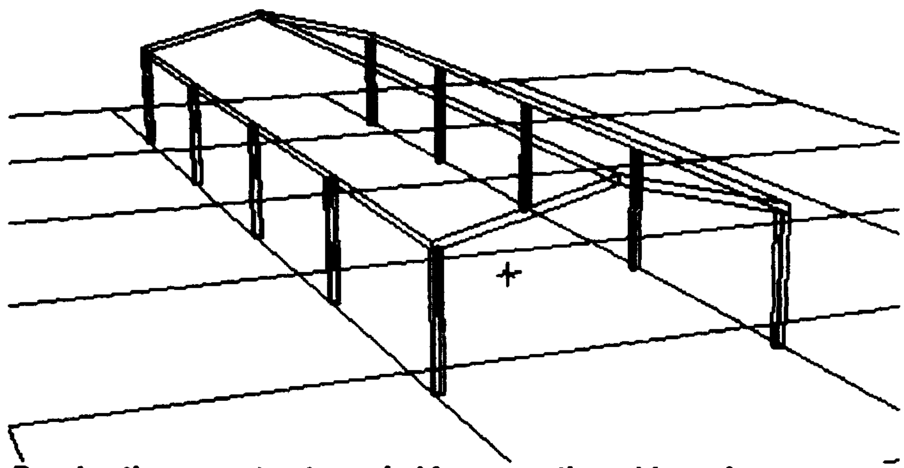
c. Click on OK and handles will appear on the visible surfaces of the shapes.

d. Select a handle on one of the columns. The shape will be duplicated four times and spaced 16.375 feet apart.

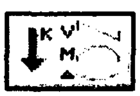
e. Select the handle on the other column and it will be duplicated four times, spaced 16.375 feet apart.

f. Double click the right mouse key to exit the Duplicate Shapes command.

9. This completes creation of the model.



F. Develop the open structure wind forces on the gable roof.

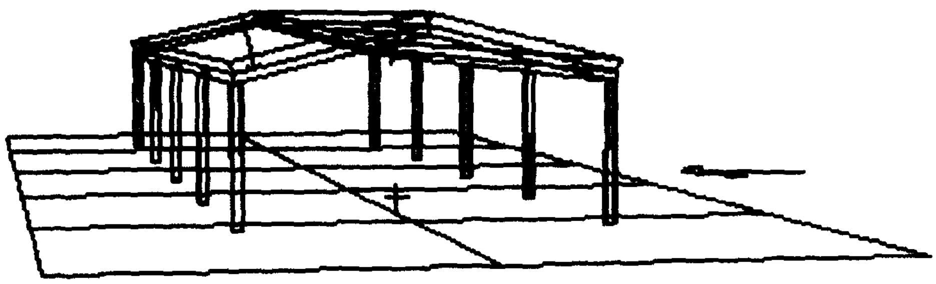


1. Select the LOADS AND DESIGN tool palette.

» **Note: A 3-D view must appear on the screen to be able to calculate open roof wind loads.**

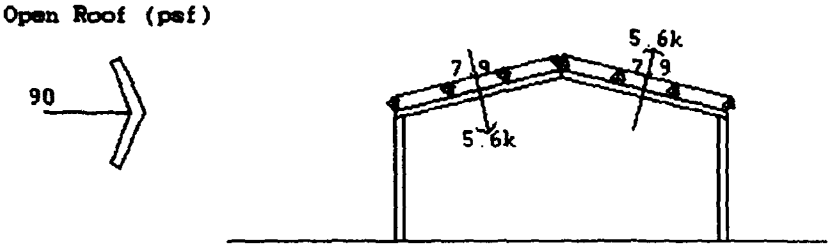
2. Select WIND from the Loads pull-down menu. A Wind Loads dialog window will appear.
3. Verify values in the Wind Loads dialog window and turn on OPEN ROOF. Modify any values as desired.
4. Click on OK for CASM to begin finding the Open Roof planes. Handles will appear on the open planes found.
5. Use the mouse pointer to select one of the gable roof planes to receive wind loads. Wind calculations are then performed on both open gable planes. A 3-D depiction of the wind load will appear on the model when calculations are completed.

G. Manipulate the building model and its wind loads.



H. Generate hard copies.

Wind Load: Open Roof (psf)



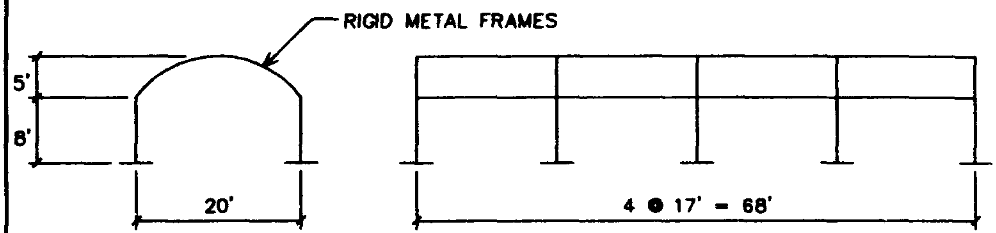
Unenclosed Building Example 2 sample output:

```
Project      : Carport
Location     : Chanute AFB - Rantoul
Design Load  : TM 5-809-1 1986
Time         : Fri Sep 13, 1991   9:46 AM

******************************** Wind Load ********************************

Velocity  Importance  Exposure   Width     Length              Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                          (ft)      (ft)
----------------------------------------------------------------------------
  70.0      1.00        C        68.0      10.0                 Gable
                                                          WW :  3.00 in 12
Distance to ocean line >= 100 mi.                         Lee:  3.00 in 12
************************ Open Roof Pressures (psf) ************************

z = h = 9.25 ft
Gh = 1.32
Kz = 0.80
qz = 0.00256*Kz*(I*V)*(I*V) = 10.00 psf
Af = [L/cos(theta)]*B = 700.8 sqft
theta = 14.0 deg < 30 deg
Cf = 0.6
F = qz*Gh*Cf*Af
+---------------------------+
|   Windward F =    5.55 k  |
+---------------------------+
|   Leeward  F =   -5.55 k  |
+---------------------------+
X = 0.5*L = 5.00 ft
w = F/Af
+---------------------------+
|  Windward w =    7.92 psf |
+---------------------------+
|  Leeward  w =   -7.92 psf |
+---------------------------+

Notes for open roof pressures:
   Positive pressures act toward surfaces.
```

I. Save the building model with its wind loads applied for future reference.

EXAMPLE THREE: One-story, Open arched roof

Given: The one-story open arched roof carport shown below. All criteria are the same as Example 2.



Required: Determine the design wind pressures on the roof.

Solution: The source for exterior force coefficients is the same as Example 2.

A. Establish Criteria.

1. Input the following data into the PROJECT, REGIONAL, and SITE Criteria dialog windows:

PROJECT:	Project Name	: Carport
	City/Installation	: Chanute AFB - Rantoul
	State	: IL
	Design Load	: TM 5-809-1 1986
REGIONAL:	Basic Wind Speed	: 70 mph
	Coastal	: No
SITE WIND:	Importance	: Category I
	Exposure	: Category C
	Distance to Oceanline	: 100 mile

B. Draw volumetric model.

1. Select the DRAW MODEL tool palette.
2. Establish general layout requirements which are different than previously established.
   a. Use the following:

SNAP INCREMENT			: 3 inches
SNAP TO UNITS			: on
SHOW GROUND PLANE			: on
GROUND PLANE			
	WIDTH	N-S	: 100 feet
		E-W	: 100 feet
	SPACING	N-S	: 20 feet
		E-W	: 20 feet
INITIAL SHAPE SIZE			
	N-S WIDTH		: 20 feet
	E-W WIDTH		: 68 feet

HEIGHT	: 5 feet
PLANE THICKNESS	: 6 inches
COLUMN WIDTH	: 6 inches
ORIENTATION	: E-W
STACK ON GROUND PLANE	: on
DIRECTIONS LOCKED	: none

» **Note: Height refers to the crown height of the arch for this example.**

3. Select OPEN BARREL VAULT from the Shapes pull-down menu. An open barrel vault will appear on the ground plane, and a Dimensions dialog window will also appear.
   a. Hold down the right mouse key while dragging the mouse toward and away from you to elevate the vault a TRANSLATED DISTANCE vertically of 8.0 feet.
   b. Click the left mouse key to fix the shape at that location. Another open barrel vault will appear on the ground plane ready for positioning.
   c. Double click the right mouse key to exit the open barrel vault command.
4. Insert the first column in the geometric model.

   a. Change the INITIAL SHAPE SIZE to reflect the column height of 8.0 feet and click on OK.

   b. Select COLUMN from the Shapes pull-down menu and place the column in the southwest corner of the roof shape.

   c. Use TAPE MEASURE to select the two vertices between which measurements are to be taken to locate the position of the column.

» **Note: Switch to the 2-D plan view to verify that the correct vertices on the roof and column have been selected, and then switch back to the 3-D view.**

   d. Use MOVE SHAPE to place the column directly under the southwest corner of the roof form. The N-S and E-W distances in the Measure dialog window should be 0.0 foot.
   e. CANCEL measuring distances between the two distances set with Tape Measure.
5. Duplicate the remaining columns.

   a. Select DUPLICATE SHAPE from the Edit pull-down menu. A Duplicate Shape dialog window will appear.
   b. Enter the following data:

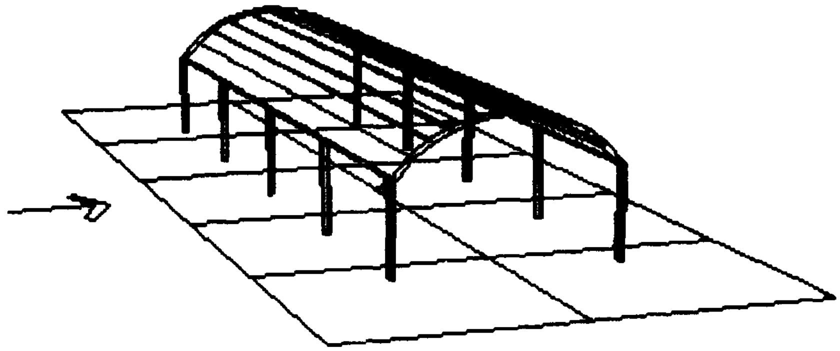
N-S DIRECTION	: 1
E-W DIRECTION	: 4
VERTICAL	: 0
N-S SPACE	: 19.0 feet
E-W SPACE	: 16.375 feet
VERTICAL SPACE	: 0.0 feet

c. Click on OK and handles will appear on the visible surfaces of the shapes.

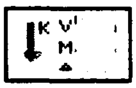
d. Select a handle on the column. The shape will be duplicated nine times, spaced at 16.375 feet apart in the E-W direction and 19.0 feet in the N-S direction.

e. Double click the right mouse key to exit the Duplicate Shape command.

6. This completes creation of the model.



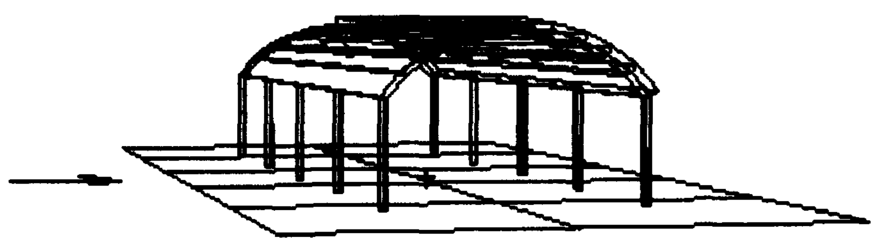
C. Develop the open structure wind forces on the open barrel vault roof.



1. Select the LOADS AND DESIGN tool palette.

>> **Note: A 3-D view must appear on the screen to be able to calculate open roof wind loads.**

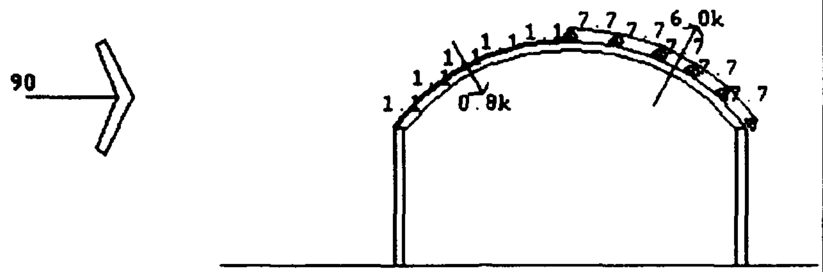
2. Select WIND from the Loads pull-down menu. A Wind Loads dialog window will appear.
3. Verify values in the Wind Loads dialog window and turn on OPEN ROOF. Modify any values as desired.
4. Click on OK for CASM to begin finding the Open Roof planes. Handles will appear on the open planes found.
5. Use the mouse pointer to select one of the planes of the polygonal construction of the barrel roof to receive wind loads. Wind calculations are then performed on all of the planes comprising the barrel vault. A 3-D depiction of the wind load will appear on the model when calculations are completed.



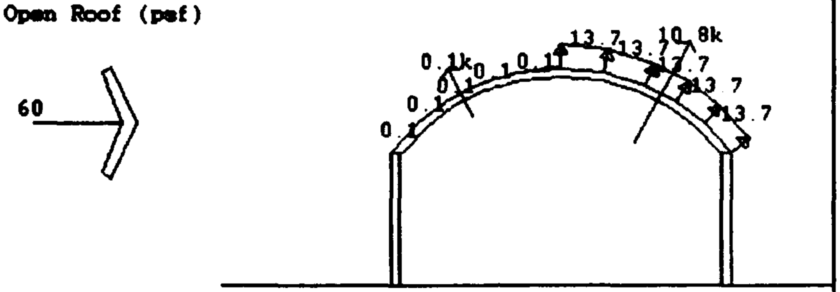
D. Manipulate the building model and its wind loads.

» **Note: Analysis has been performed for a 90- and a 60-degree wind angle. Use SHOW LOADS to display each wind load case.**

Wind Load: Open Roof (psf)



Wind Load: Open Roof (psf)



E. Generate hard copies.

Unenclosed Buildings Example 3 sample output:

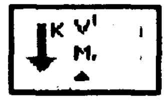
```
Project     : Carport
Location    : Chanute AFB - Rantoul
Design Load : TM 5-809-1 1986
Time        : Fri Sep 13, 1991   9:57 AM

******************************** Wind Load ********************************

Velocity  Importance  Exposure   Width     Length              Roof Type
            Factor              Perpend.  Parallel
                                to Wind   to Wind
  (mph)                           (ft)      (ft)
---------------------------------------------------------------------------
   70.0     1.00         C        68.0      20.0                  Arched
                                                          Crown:   5.0 ft
Distance to ocean line >= 100 mi.

************************ Open Roof Pressures (psf) ************************

z = h = 10.50 ft
Gh = 1.32
Kz = 0.80
qz = 0.00256*Kz*(I*V)*(I*V) = 10.00 psf
f/L = 0.25
Af = 788.20 sqft

90 deg wind
Windward half Cf = +0.08
Leeward half  Cf = -0.58
F = qz*Gh*Cf*Af
+----------------------------+
|   Windward F =   0.83 k    |
+----------------------------+
|   Leeward  F =  -6.03 k    |
+----------------------------+
w = F/Af
```

```
+----------------------------+
|  Windward w =    1.06 psf  |
+----------------------------+
|  Leeward  w =   -7.66 psf  |
+----------------------------+

60 deg wind
Windward half Cf = -0.01
Leeward half  Cf = -1.04
+----------------------------+
|   Windward F =   -0.10 k   |
+----------------------------+
|   Leeward  F =  -10.82 k   |
+----------------------------+
w = F/Af
+----------------------------+
|  Windward w =   -0.13 psf  |
+----------------------------+
|  Leeward  w =  -13.73 psf  |
+----------------------------+

Notes for open roof pressures:
   Positive pressures act toward surfaces.
```

F. Save the building model with its wind loads applied for future reference.

DEAD LOADS

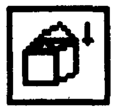
This section describes the technique that CASM employs to generate dead loads for floor, roof, ceiling, and wall assemblies you may design. SNOW and WIND load design, as you have already seen, is dependent on data input into the three CRITERIA windows. The DEAD LOADS program, however, is independent of the CRITERIA menu and its associated pop-up dialog windows. A volumetric model does not need to be drawn to create dead load assemblies.



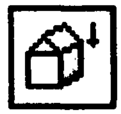
Select the LOADS AND DESIGN Tool Palette. You may then proceed to LOADS on the Loads and Design menu bar and scroll down to ROOF (DL), FLOOR (DL), CEILING (DL), or WALL (DL). The other option is to select either of the four respective icons from the Loads and Design Tool Palette.

ROOF

FLOOR



CEILING



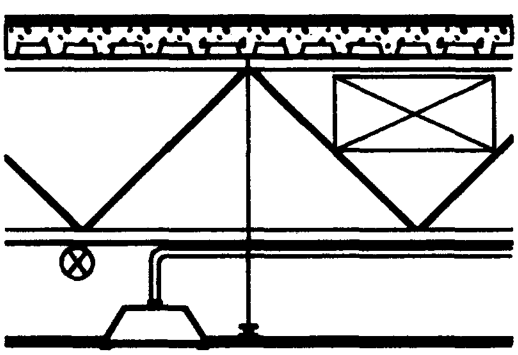
WALL

Regardless of the option preferred, click the left mouse key on the desired Dead Load and highlight it. The accompanying pop-up dialog window will appear, and you are ready to develop the material weights for the assembly. Many system assemblies can be generated for a given project. They can be stored and retrieved as needed. The following examples will take you step by step through dead load calculations for wood, steel, and concrete floor and roof assemblies.

■ Floor Assemblies

EXAMPLE ONE: Open-web steel joist framing

Given: The floor assembly shown:

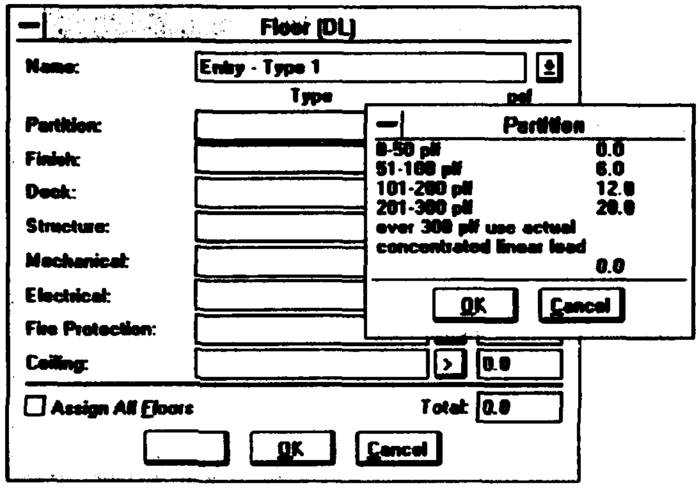


Required: Calculate the total assembly dead load and save as ENTRY-TYPE 1

Solution:

1. Select LOADS from the top menu bar and scroll to FLOOR (DL) or select the FLOOR (DL) ICON from the tool palette. A FLOOR (DL) pop-up dialog window will appear.
2. Type 'Entry-Type 1' over the highlighted current name.
3. Input the assembly materials.

   a. Move the mouse pointer to a required data window button.
   b. Click the left mouse key and a pop-up dialog window will appear showing a list of possible materials.



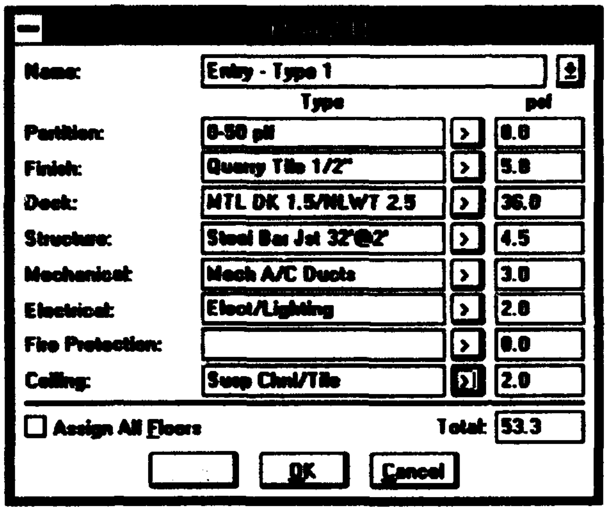
   c. Scroll the pop-up dialog window list for the desired material.
   d. Place the mouse pointer on that material and click the left mouse key to highlight your choice.
   e. Click on OK. Your material choice and its corresponding weight will appear in the 'Type' box and 'psf' box. The 'Total' box will automatically sum the weights of all current choices.

» **Note: An alternate approach is to select the material by double clicking on the highlighted material. This avoids having to also click on OK.**

   f. Complete the filling in of all 'Type' boxes as follows:

Partition	:	0 - 50 PLF
Finish	:	1/2" quarry tile
Deck	:	MTL DK 1.5/ NLWT 2.5
Structure	:	Steel bar jst. 32'@ 2'
Mechanical	:	Mech a/c ducts (3 psf)
Electrical	:	Elect/ Lighting (2 psf)
Fire Protection	:	none required
Ceiling	:	Susp. Chnl./ Tile

Upon completion of all entries a total weight of 53.3 psf will exist in the 'total' box. You can edit or change any item in a 'type' or 'psf' box, as described in the REFERENCE chapter of the Reference Manual. The FLOOR (DL) window should look as follows:



» Floor assemblies are automatically saved for each unique floor name. Use the Floor Name drop-down list to view and edit other floor assemblies.

4. Select OK after you completed entering floor assemblies to return to the CASM program window.

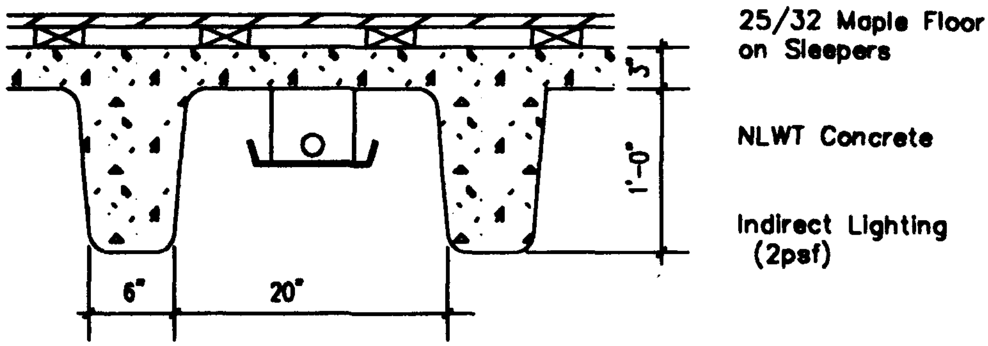
The other two options in the Floor (DL) dialog window will be addressed later. Briefly they are:

- Select Assign All Floors to apply the load to all floor planes when using the ASSIGN option.
- Select CANCEL to not save the changes to the current floor name.
- Select ASSIGN to place the current floor DL shown in the window on a floor plane in the building model. The next chapter will perform this operation.

» All the assemblies will be printed at the end of this section.

Example Two: Cast-in-place concrete pan joists

Given: The floor assembly shown:



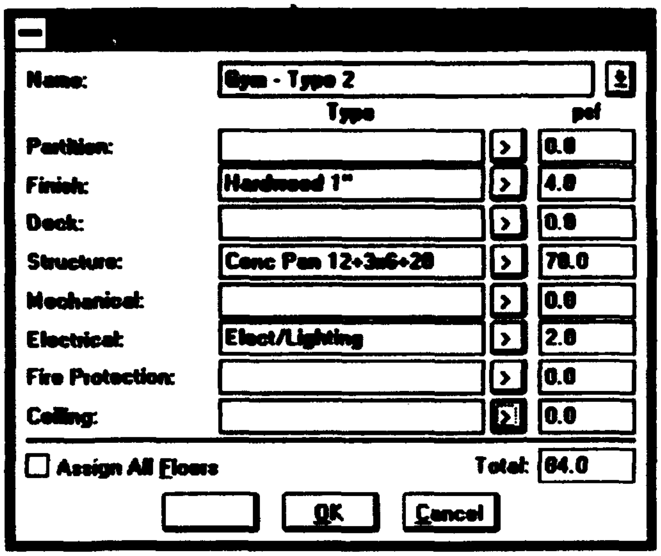
Required: Develop the total dead weight of the given assembly and name it GYM-Type 2.

Solution:

This example proceeds the same as Example 1, except none of the components exist in the materials database, and you must create them.

1. Select FLOOR (DL) from the Loads pull-down menu in the Loads and Design tool palette. The last entered assembly will appear on the FLOOR (DL) pop-up dialog window.
2. Type over the present NAME and enter: GYM-TYPE 2.
3. Delete the PARTITION 'TYPE' and 'PSF.'
   a. Select the Partition data window button. The pop-up dialog window of materials will appear.
   b. Scroll the pop-up dialog window for the blank 0.0.
   c. Place the mouse pointer over that selection and click the left mouse key to highlight your choice.
   d. Click on OK. The partition type will become blank and 0.0 psf will be inserted.
4. Replace the FINISH material and weight with 1" hardwood.
5. Repeat step 3 for DECK to clear the 'TYPE' box and place 0.0 in the 'PSF' box.
6. Replace the STRUCTURE material and weight with the concrete pan joist proportions given. Scrolling the choices in the pop-up dialog window reveals that the closest choice is Conc Pan 12+3x5+20 weighing 74.0 psf. Select it and edit the 'TYPE' and 'PSF' boxes to read: CONC PAN 12+3x6+20 weighing 78 psf. An alternate approach would have been to merely write in the entire description of the item and its new weight.

7. Delete MECHANICAL 'TYPE' description and place 0.0 in the 'PSF' box as in step 3.
8. Leave the ELECTRICAL boxes as they are.
9. Delete CEILING 'TYPE' description and place 0.0 in the 'PSF' box as in step 3. The new assembly FLOOR (DL) window will look as follows:



>> You will select ASSIGN to place the current floor DL shown in the window on a floor plane in the building model later. The next chapter will perform this operation.

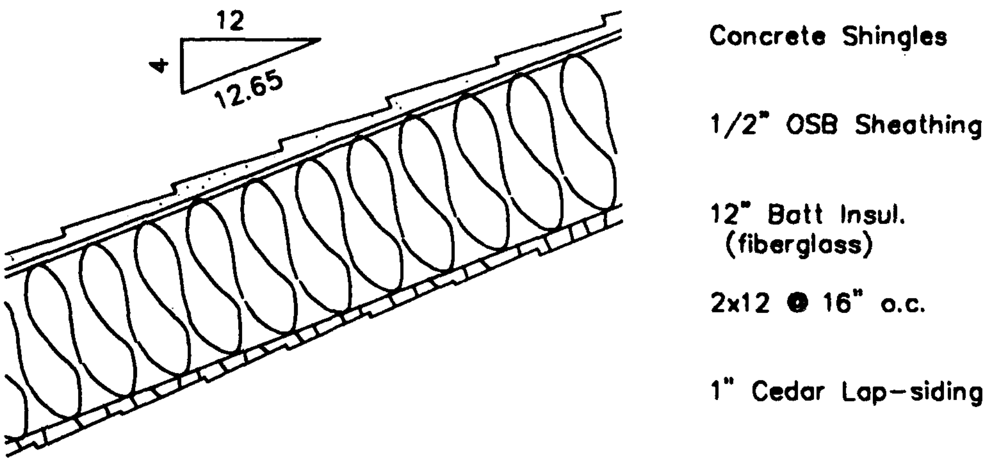
10. Use the Name drop-down list to review all the saved FLOOR (DL) assemblies for your project.
11. Edit either assembly, or select OK to end the development of floor dead load types.

>> The hard copy will be printed at the end of this section.

■ Roof Assemblies

EXAMPLE ONE: Wood rafter framing

Given: The wood framing section shown:



Required: Calculate the system dead weight and save as HOUSE-TYPE 1.

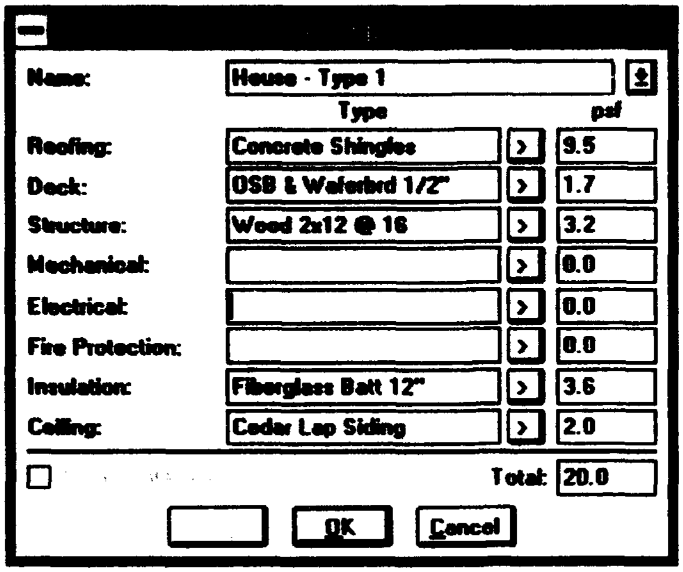
Solution:

It is assumed that you now have an understanding of the process and it is assumed that you can bring up the ROOF (DL) pop-up dialog window and enter the new name.

1. Enter the following ROOF (DL) 'TYPE' and 'PSF' items:

Name	:House - Type 1	
Roofing	:Concrete shingles	9.5
Deck	:OSB & Waferbrd 1/2"	1.7
Structure	:2x12 @ 16"	3.2
Insulation	:Fiberglass batt 12"	3.6

2. Scroll the ceiling choices and select CANCEL since Cedar Lap Siding is not listed. Type in the material and its 2.0 psf weight.
3. Select SAVE to store the assembly. The completed dialog window looks as follows:



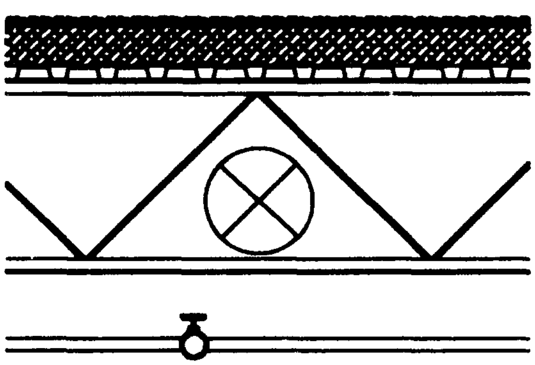
» You will select ASSIGN in the next chapter to place the current roof DL shown in the window on a roof plane in the building model

4. The total weight of this assemblage of components is 20.0 psf acting downward along the slope.

» Note: The computer will calculate the projected load value during analysis.

[illegible] joist framing

Given: The steel joist framing section shown:



Ballasted Single-Ply Roofing

Rigid Tapered Insul 3" – 7" (use 5" average)

1 1/2" Metal Deck – 20 ga

Metal Ducts: 1 psf

24' Span Joists @ 4'-0" o.c.

Electrical: 1 psf

Wet Sprinkler System: 2 psf

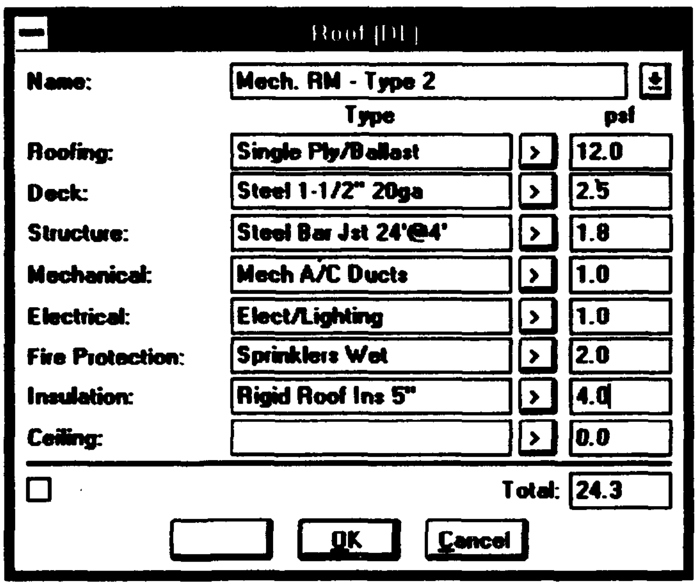
Required: Calculate the total dead weight of the given assembly and name it MECH.RM-TYPE 2.

Solution:

1. Edit the previous ROOF (DL) window with the following data:

Name	: Mech.RM-Type 2	
Roofing	: Single-ply/ Ballast	12.0
Deck	: Steel 1-1/2 - 20 ga	2.5
Structure	: Steel bar jst. 24'@ 4'	1.8
Mechanical	: Mech a/c ducts	1.0
Electrical	: Elect/ Lighting	1.0
Fire protection	: Sprinklers - wet	2.0
	(must edit the 'PSF' box to change the 0.8 to 2.0)	
Insulation	: Rigid roof insul. 5"	4.0
	(must change the database value of 6 to 5 and 4.8 to 4.0)	
Ceiling	: none	

2. The TOTAL system weight of 24.3 psf is shown. A sample ROOF (DL) window follows:



» You will select ASSIGN in the next chapter to place the current roof DL shown in the window on a roof plane in the building model.

3. Select the drop-down list button to review all the saved ROOF (DL) assemblies.

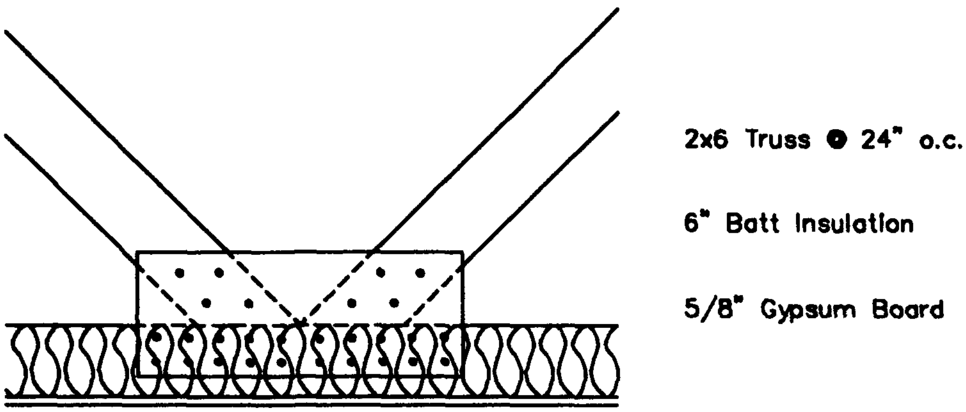
4. Select OK to return to the cleared CASM program window.

■ Ceiling Assemblies

Many times in the design of trusses it is advantageous to separate top chord and bottom chord dead loads and live loads. This is typical in wood, but also in steel trusses for floors or roofs. Interstitial trusses, used most often in hospitals, also require separation of top and bottom chord loads; however, since these ceiling assemblies also combine with flooring, they are best treated as a FLOOR (DL). Thus, they are the truss exception. A typical example of a CEILING (DL) would thus be the bottom chord of a wood truss.

EXAMPLE ONE: Wood roof truss system

Given: The bottom chord of a metal plate connected wood truss for a residential application.



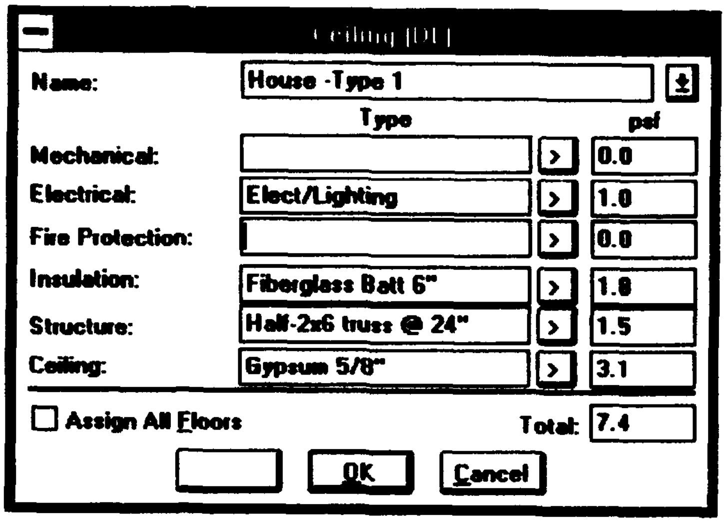
Required: Prepare the bottom chord dead load for the truss fabricator and his engineer.

Solution:

1. Bring up the CEILING (DL) pop-up dialog window and input the following data:

Name	: House-Type 1	
Mechanical	: none	0.0
Electrical	: Elect/ Lighting	1.0
Fire Protection	: none	0.0
Insulation	: Fiberglass batt 6"	1.8
Structure	: Half- 2x6 truss @ 24"	1.5
Ceiling	: Gypsum 5/8"	3.1

2. A completed window should look as follows:



» You will select ASSIGN in the next chapter to place the current ceiling DL shown in the window on a floor plane in the building model.

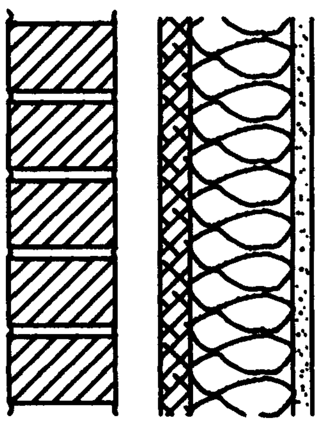
3. Select OK to return to the cleared CASM program window.

Wall Assemblies

The weight of wall assemblies is prepared in a similar fashion to that of floors, roofs, and ceilings. The wall weights will be in PSF and must merely be multiplied by the wall height to obtain linear loads in PLF, or merely multiplied by the wall area to obtain the total wall weight in pounds. When the load is assigned, the computer will calculate the linear load of the assembly.

[illegible] Wall Brick Veneer

Given: Wall section shown below:



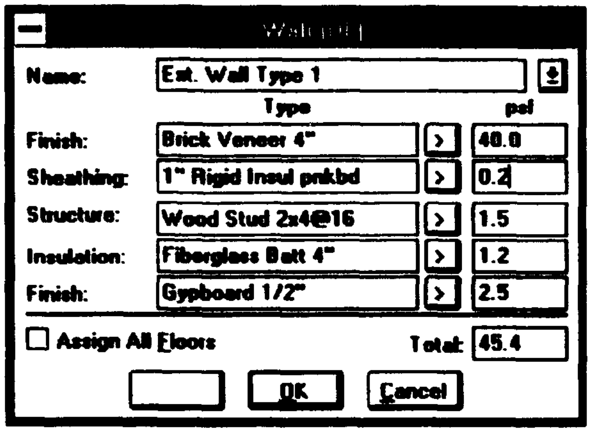
Required: Determine wall weight in PSF and name it EXT.WALL-TYPE 1.

Solution:

1. Select WALL (DL) from the Loads pull-down menu or from the icon on the Loads and Design tool palette. The WALL (DL) pop-up dialog window will appear.
2. Enter the following 'TYPE' and 'PSF' items:

Name	: Ext.Wall-Type 1	
Finish	: Brick veneer 4"	40.0
Sheathing	: 1" rigid insul. (pinkbrd)	0.2
Structure	: 2x4 @ 16"	1.5
Insulation	: Fiberglass batt 4"	1.2
Finish	: Gypboard 1/2"	2.5

3. A completed window should look as follows:



>> You will select ASSIGN in the next chapter to place the current wall DL shown in the window on a floor plane in the building model. You would select Assign All Floors to apply the load to all floor planes when using the ASSIGN option.

>> The next wall assembly can be prepared by directly changing values in the current WALL (DL) assembly window. It is necessary to insert a new name, unless it is desired to just replace the existing assembly.

4. Select OK to return to the cleared CASM program window.
5. Obtain a hard copy of the assemblies.

   a. Select File from the CASM top menu bar and scroll down to PRINT DATA. The *Print Data* pop-up dialog window will appear.
   b. Place an 'X' in the DEAD & LIVE LOADS option.
   c. Remove the 'X' in all other output options. This will only print the Loads assemblies.
   d. Place an 'X' in the box for either PRINT TO PRINTER or PRINT TO FILE.
   e. Click on OK and if you selected PRINT TO PRINTER, your printer will be activated. If you selected PRINT TO FILE, you will be placed in NOTEPAD.

>> Note: See the Printing Project Criteria Data in this Tutorial Manual for more information on the PRINT DATA command.

Sample Loads Output:

Loads

Floor Dead Loads

Name : Entry-Type 1

	Type	psf
Partition	: 0-50 PLF	0.0
Finish	: Quarry Tile 1/2"	5.8
Deck	: MTL DK 1.5/NLWT 2.5	36.0
Structure	: Steel Bar Jst 32'@2'	4.5
Mechanical	: Mech A/C Ducts	3.0
Electrical	: Elect/Lighting	2.0
Fire Protection	:	0.0
Ceiling	: Susp Chnl/Tile	2.0
Total	:	53.3

Name : Gym-Type 2

	Type	psf
Partition	:	0.0
Finish	: Hardwood 1"	4.0
Deck	:	0.0
Structure	: Conc Pan 12+3x6+20	78.0
Mechanical	:	0.0
Electrical	: Elect/Lighting	2.0
Fire Protection	:	0.0
Ceiling	:	0.0
Total	:	84.0

Roof Dead Loads

Name : House-Type 1

	Type	psf
Roofing	: Concrete Shingles	9.5
Deck	: OSB & Waferbrd 1/2"	1.7
Structure	: Wood 2x12 @ 16	3.2
Mechanical	:	0.0
Electrical	:	0.0
Fire Protection	:	0.0
Insulation	: Fiberglass Batt 12"	3.6
Ceiling	: Cedar Lap Siding	2.0
Total	:	20.0

Name : Mech.RM-Type 2

	Type	psf
Roofing	: Single Ply/Ballast	12.0
Deck	: Steel 1-1/2" 20ga	2.5
Structure	: Steel Bar Jst 24'@4'	1.8
Mechanical	: Mech A/C Ducts	1.0
Electrical	: Elect/Lighting	1.0
Fire Protection	: Sprinklers Wet	2.0
Insulation	: Rigid Roof Ins 5"	4.0
Ceiling	:	0.0
Total	:	24.3

Ceiling Dead Loads

Name : House-Type 1

	Type	psf
Mechanical	:	0.0
Electrical	: Elect/Lighting	1.0
Fire Protection	:	0.0
Insulation	: Fiberglass Batt 6"	1.8
Structure	: Half 2x6 Truss @ 24"	1.5
Ceiling	: Gypsum 5/8"	3.1
Total	:	7.4

Wall Dead Loads

Name : Ext.Wall-Type 1

		Type	psf
Finish	:	Brick Veneer 4"	40.0
Sheathing	:	1" Rigid Insul pnkbd	0.2
Structure	:	Wood Stud 2x4@16	1.5
Insulation	:	Fiberglass Batt 4"	1.2
Finish	:	Gypboard 1/2"	2.5
Total	:		45.4

MINIMUM ROOF LIVE LOAD

The basic minimum roof live load for members supporting flat, pitched, or arched roofs is 20 psf as prescribed in ANSI-A58.1-1982 and cited in TM 5-809-1 1986 referenced at the beginning of this chapter. Reductions to the 20 psf are a function of the horizontal projected tributary area carried by the member and the roof slope. The following example illustrates this provision as set up in CASM.

[illegible]

Given: An industrial building framed with repetitive bays at 20 feet on center. The roof is framed with 60-foot span trusses in each bay. The top chord slope is 3 in 12 creating a low sloped gable roof. This example can be found in TM 5-809-1 1986 on page C-2.

Required: Determine the minimum live load to be carried by each truss.

Solution:

A. Establish Criteria.

1. Input the following data into the PROJECT dialog window:

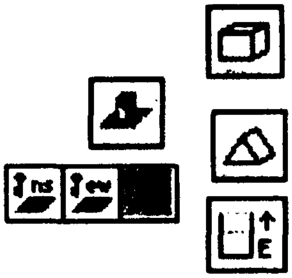
PROJECT:	Project Name	: Industrial Building
	City/Installation	: Vicksburg
	State:	: MS
	Design Load	: TM 5-809-1 1986

» **Note: This is the minimum information required to do minimum roof live load calculations.**

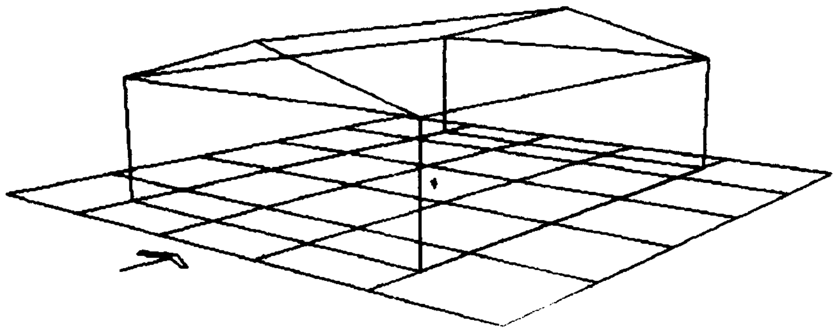
B. Draw volumetric model.

1. Select the DRAW MODEL tool palette.
2. Establish general layout requirements which are different than previously established.
   a. Use the following:

DEFINE UNITS(snap increment): 6 inches			
SNAP TO UNITS			: on
SHOW GROUND PLANE			: on
GROUND PLANE			
	WIDTH	N-S	: 100 feet
		E-W	: 100 feet
	SPACING	N-S	: 20 feet
		E-W	: 20 feet
INITIAL SHAPE SIZE			
	N-S		: 80 feet
	E-W		: 60 feet
	HEIGHT		: 20 feet
	ORIENTATION		: N-S
STACK ON GROUND PLANE			: on
DIRECTIONS LOCKED			: none

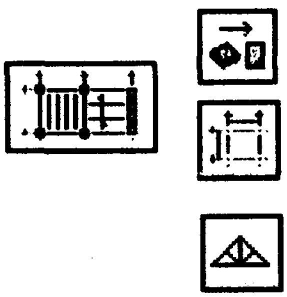


3. Place a CUBE on the ground plane with the required dimensions.
4. Draw the gable roof.
   a. Turn on STACK ON LAST SHAPE.
   b. Stack a PRISM on the cube.
   c. LOCK the N-S and E-W directions.
   d. Use the DRAG EDGE command to make the roof slope 3 in 12.
   e. UNLOCK the N-S and E-W directions.



5. Draw roof structural elements.

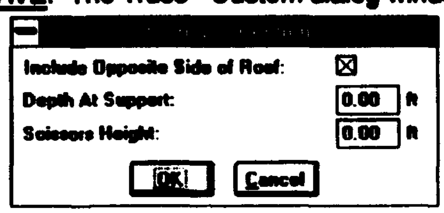
» If unfamiliar with drawing structure, refer to Wind Load Example 1, step 6, Floor Framing Scheme 1a step E-4, or the CASM Reference Manual.



   a. Select INCLINED STRUCTURAL PLANE.
   b. Select DRAW STRUCTURE tool palette.
   c. Select DEFINE GRID from the Grid/Open pull-down menu.

      N-S Spacing :20 feet
      E-W Spacing :99 feet

   d. Select TRUSS - CUSTOM from the Surf/Line pull-down menu.
   e. Define area to draw trusses by selecting handles in clockwise order to define the entire area of the inclined plane.
   f. Set the truss Spacing to 20 feet.
   g. Set the Orientation to E-W.
   h. Turn OFF Draw Surface.
   i. Click on SAVE. The Truss - Custom dialog window will appear.



   j. Turn on INCLUDE OPPOSITE ROOF. Leave depth and scissors heights at 0.0 feet.
   k. Click on OK.

6. This completes creation of the model and insertion of roof trusses.

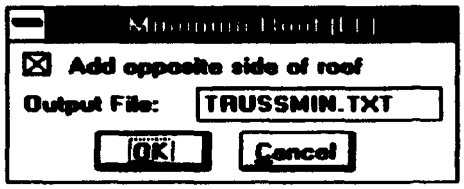
C. **Determine the minimum roof live load for a typical roof truss.**

» **Note: You must be in a 2-D view of a selected plane on the 3-D model to be able to apply a minimum roof live load. You cannot be in a 3-D, plan, elevation, or section view.**

1. Select the LOADS AND DESIGN tool palette.

» **Note: This is a view of the true width, not a view of the projected width.**

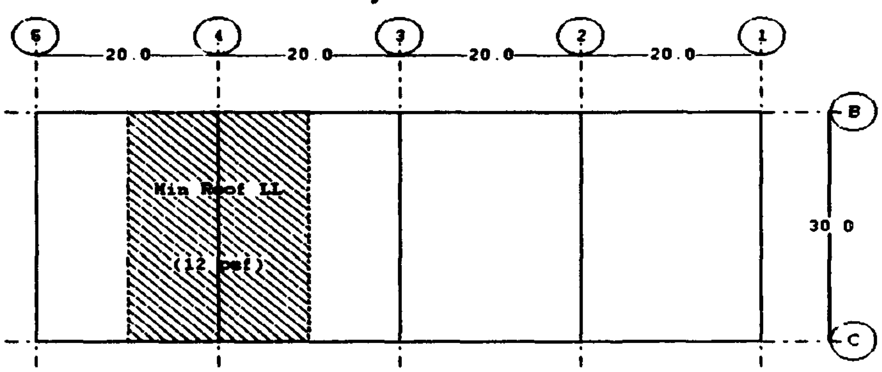
2. Select MIN. ROOF (LL) from the Loads pull-down menu. A Minimum Roof (LL) dialog window will appear.



3. Turn on ADD OPPOSITE SIDE OF ROOF.
4. Modify the Output File name to TRUSSMIN.TXT and click on OK. A Tributary Area dialog window will appear.
   a. Use the mouse pointer to set the lower left corner of the tributary area to be created. Place 10 feet from the left edge and click the left mouse key.
   b. Drag the mouse pointer to create a tributary area with a tributary width of 20 feet and a length equal to the width of the roof plane.

» **Note: The tributary area is a projected area above the 2-D plane.**

   c. Click the left mouse key to fix the tributary area. Calculation then begins. The minimum roof live load and its name will appear on the 2-D plane within the drawn tributary area.



5. Click on CANCEL to exit the Minimum Roof Live Load command.

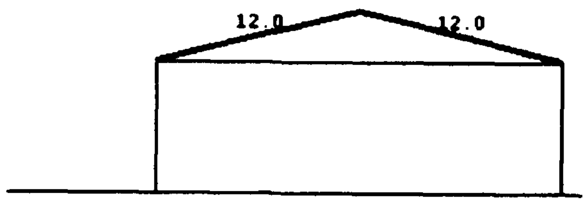
D. **Manipulation of the building model and its loads.**

1. For details on the following abbreviated commands, refer to steps D-1 through D-7 in Snow Load Example 1.
   a. Zoom the graphics on the screen.

b. Pan the screen image
c. View → Perspective 3D
d. View → Solid Object
e. Rotate 3D view
f. Adjust the viewing height.
g. Adjust the viewing distance.

E. Generation of hard copies.

Minimum Roof Live Loads (psf)



1. Print a 2-D section and calculations. For details on the following abbreviated commands, refer to steps E-1 through E-4 of Snow Load Example 1.
   a. View → Section
   b. File → Print Screen
      ✔ Printer
      OK
   c. File → Print Data
      ✔ Min Roof LL
      ✔ Print to File
      ✔ Execute Notepad
      OK
   d. Notepad → File → Page Setup
      Left Margin : 0.5
      Right Margin : 0.0
      OK
   e. Notepad → File → Print
   f. Notepad → File → Exit

Example 1 Sample Output :

```
Project     : Industrial Building
Location    : Vicksburg
Design Load: TM 5-809-1 1986
Time        : Tue Jan 30, 1990  10:48 PM

********************** Minimum Roof Live Load (Lr) **********************

Tributary area (At) : 1200 sf
Roof slope      (F)  : 3.00 in 12

Lr = 20*R1*R2 >= 12
At >= 600          R1 = 0.60
F <= 4             R2 = 1.00
Lr = 12.00 psf
```

```
minimum Lr = 12 psf

+-----------------------+
|     Lr = 12.00 psf    |
+-----------------------+
Check minimum roof live load, Lr, against minimum snow design loads.

Additionally, for the design of secondary members such as roof decking and
rafters, a concentrated live load with 250 lbs uniformly distributed over
an area of 2 feet square (4 sqft) will be included. The concentrated load
will be located so as to produce the maximum stress in the member.
```

F. Save the building model with its minimum live load applied for future reference.

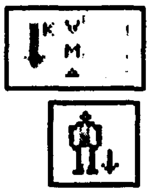
1. **Select SAVE from the File pull-down menu on the CASM menu bar.**
2. **Type in Filename: MINROOF.BLD.**
3. **Click mouse on SAVE.**

LIVE LOADS: OCCUPANCY

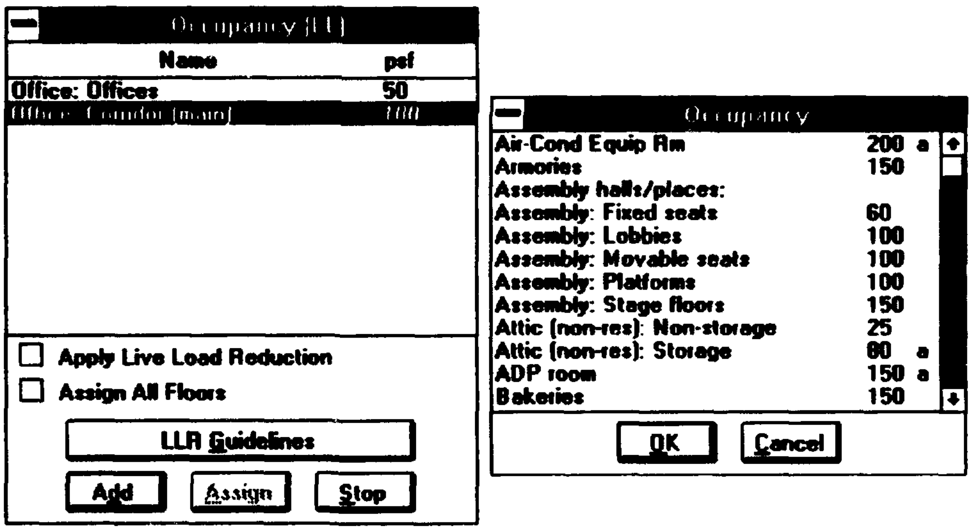
This section describes the procedures used by the CASM program to generate uniformly distributed floor occupancy live loads for different projects. This version of CASM addresses occupancy live loads based on the provisions stated in the TM 5-809-1 manual only.

It is not necessary to select a Design load code from the PROJECT CRITERIA window to prepare occupancy live loads, as it was for SNOW and WIND loads. It is not necessary to draw the geometric model to create a list of occupancy live loads for the building.

__A typical procedure for creating the occupancy live load list for the building is as follows:__

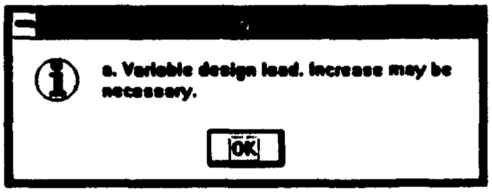


1. Select the LOADS AND DESIGN tool palette.
2. Select Loads from the menu bar and scroll down to OCCUPANCY (LL). Click on this highlighted choice with the left mouse key. The typical alternate approach is to select the occupancy live load icon from the LOADS tool palette. An OCCUPANCY (LL) pop-up dialog window will appear.

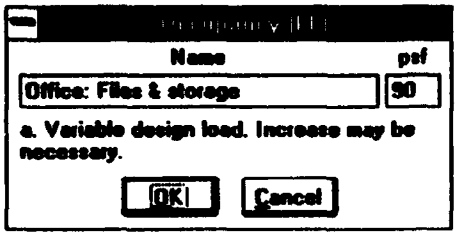


3. Drag the mouse pointer to the ADD box and click on the left mouse key. This will activate the occupancy live load selection list in the overlay pop-up dialog window.
4. Use the scroll bar arrows to scan the list for desired choices.
5. Use the mouse pointer to select a choice.
6. Double click the left mouse key or single click and then click on OK. The occupancy live load will appear in the OCCUPANCY (LL) window.

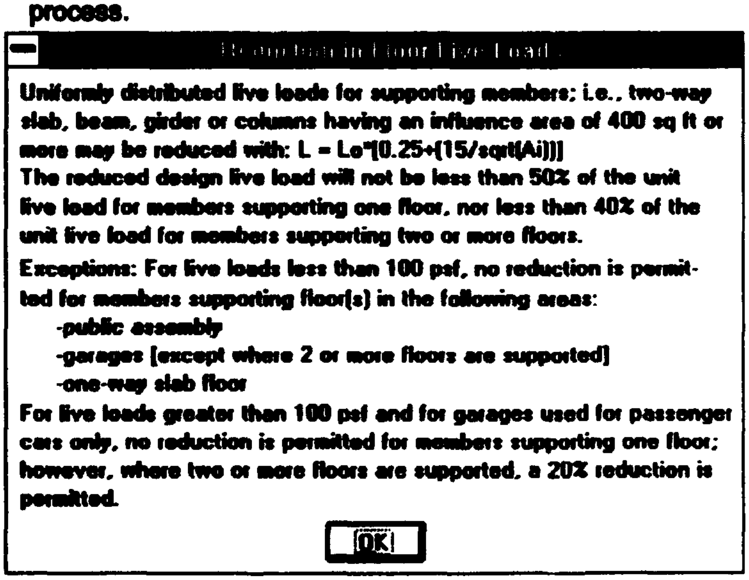
» **Note: Certain live load magnitudes are followed by a letter which represents a reminder note that appears on the screen when the load is selected. Click on OK to remove the note.**



7. To modify the live load magnitude or name, or view the note:
   a. Double click the left mouse key on the live load name. A pop-up dialog window will appear.



   b. Change the name or psf as desired.
   c. Select OK when satisfied or CANCEL to not save the change.
8. Repeat the process as many times as required to create a list of the project's uniformly distributed live loads.
9. Decide if live load reductions are to be applied to all the loads listed.
   a. Select LLR GUIDELINES for the code criteria to aid your decision process.



   b. Place an 'X' next to APPLY LIVE LOAD REDUCTION to have the reductions automatically calculated during analysis.

» **Note: It is not possible to have live load reduction apply to only a select few loads from the list.**

» **Select ASSIGN ALL FLOORS to apply the load to all floor planes when using the Assign option.**

» **Select ASSIGN to apply the highlighted live load from the list on a floor plane in the building model. This process is performed in the next chapter.**

10. When finished, select <u>STOP</u> to return to a clear CASM program window.

» **Your live load choices can be saved to a file according to procedures described in the REFERENCE and OVERVIEW chapters of the Reference Manual. The following example illustrates the application of the discussion described above.**

EXAMPLE ONE: Multiuse facility

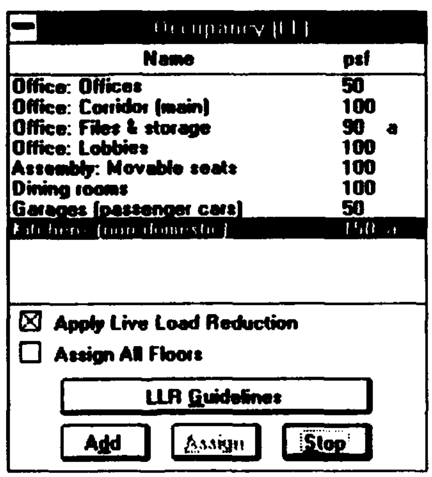
Given: A four-story multiuse facility that will be designed based on the TM 5-809-1 1986 Loads Manual contains the following functions:

Offices (3 stories)
Corridors (main)
Files and storage
Lobbies
Lecture hall, Meeting room w/ movable seats
Dining room
Parking garage (1 story)
Kitchen

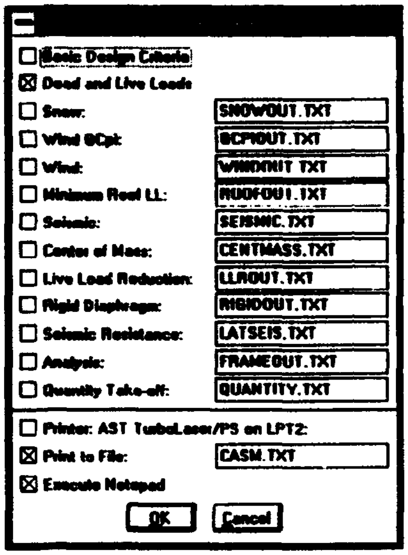
Required: Prepare a list of live loads for the project. Live load reductions are to be taken for all live loads. The Files and Storage live load should be increased to 90 psf. Save the list in a file called OFFICE.BLD.

Solution:

1. Follow the steps outlined above to create the project's live load list. The completed OCCUPANCY (LL) window will look as follows:



2. Select STOP and return to the clear CASM program window.
3. Save the project live load list to a file.
   a. Select the File pull-down menu from the CASM top menu bar and select SAVE. A pop-up dialog window will appear.
   b. Type in the project filename: OFFICE.BLD
   c. Select OK. The filename 'untitled' at the top of the CASM window will be replaced with the new filename.
4. Print the project live load list.
   a. Select File pull-down menu again and select PRINT DATA. The Print Data dialog window will appear.



   b. Select DEAD & LIVE LOADS and PRINT TO FILE.
   c. Enter the filename: OFFICE.TXT.

» Note: If you enter .BLD it will automatically be changed to .TXT for the NOTEPAD program.

   d. Turn on EXECUTE NOTEPAD.
   e. Select OK and respond to the pop-up dialog window warning to replace the output file if one appears.
   f. The NOTEPAD program window will appear displaying the live load file and accompanying notes.
   g. Select PAGE SETUP from the File pull-down menu.
      (1) Set the left margin to 0.5 and the right margin to 0.
      (2) Click on OK.
   h. Select the File pull-down menu and select PRINT. The file is then sent to be printed on the printer. A sample output is as follows:

Loads

Occupancy Live Loads

Name	psf
Office: Offices	50
Office: Corridor (main)	100
Office: Files & storage	90a
Office: Lobbies	100
Assembly: Movable seats	100
Dining rooms	100
Garages (passenger cars)	50
Kitchens (non domestic)	150a

a. Variable design load. Increase may be necessary.

Notes

Uniformly distributed live loads for supporting members; i.e., two-way *slab, beam, girder or columns having an influence area of 400 sq ft or* more may be reduced with: L = Lo*[0.25+(15/sqrt(Ai))]
The reduced design live load will not be less than 50% of the unit live load for members supporting one floor, nor less than 40% of the unit live load for members supporting two or more floors.
Exceptions: For live loads less than 100 psf, no reduction is permitted for members supporting floor(s) in the following areas:

- -public assembly
- -garages [except where 2 or more floors are supported]
- -one-way slab floor

For live loads greater than 100 psf and for garages used for passenger cars only, no reduction is permitted for members supporting one floor; however, where two or more floors are supported, a 20% reduction is permitted.

» **Note: An alternate (simpler) way to obtain a printout is to select PRINT TO PRINTER instead of PRINT TO FILE. The latter was done here merely to show the process.**

STRUCTURAL ANALYSIS AND DESIGN

This chapter is intended to present the structural planning capabilities of CASM. It will synthesize the many pieces of CASM you have learned in the previous chapters and give you an understanding of the program's application in the preliminary structural design process. The entire flowchart, illustrated in Chapter 1, will now be used to compare structural systems and assist the engineer in his decision-making process.

This chapter will assume that you have mastered CRITERIA, LOADS generation, and the basics of GEOMETRIC MODELING from the previous chapters. Emphasis will be on the commands necessary to:

A. Establish structural grids
B. Create openings
C. Draw structural framing systems
D. Establish structural element parameters
E. Assign loads and generate load combinations
F. Perform preliminary analysis
G. Perform preliminary structural member design

■ Floor Framing Design Comparison

EXAMPLE ONE: Alternative structural schemes for a repetitive floor framing system

Given: A three-story 6 x 3 bay office building. A typical bay will be 24 feet by 24 feet. The building will be a braced frame with x-bracing around the corner stair towers providing lateral load resistance.

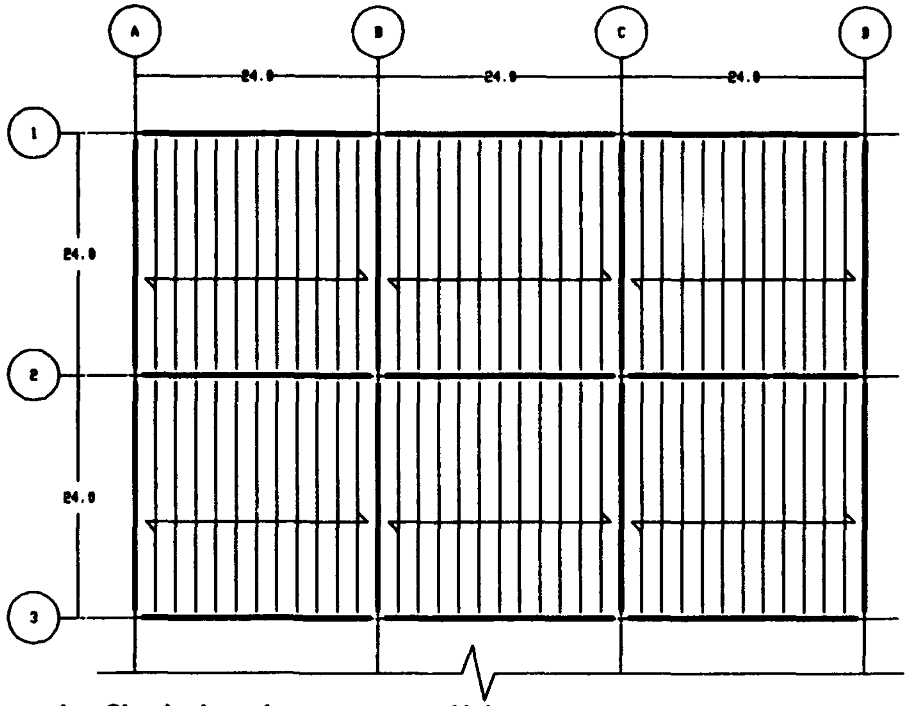
The occupancy live load will be assumed a smear of office, corridor and partitions totaling 70 psf.

The exterior wall construction will include 4-inch brick veneer with a 2-inch air space and a lightweight 8-inch CMU back-up. A 1-inch rigid insulation (expanded polystyrene) will be placed in the cavity. The exterior wall will be supported at each floor level. The exterior face of brick is 9 inches in front of the spandrel beam centerline.

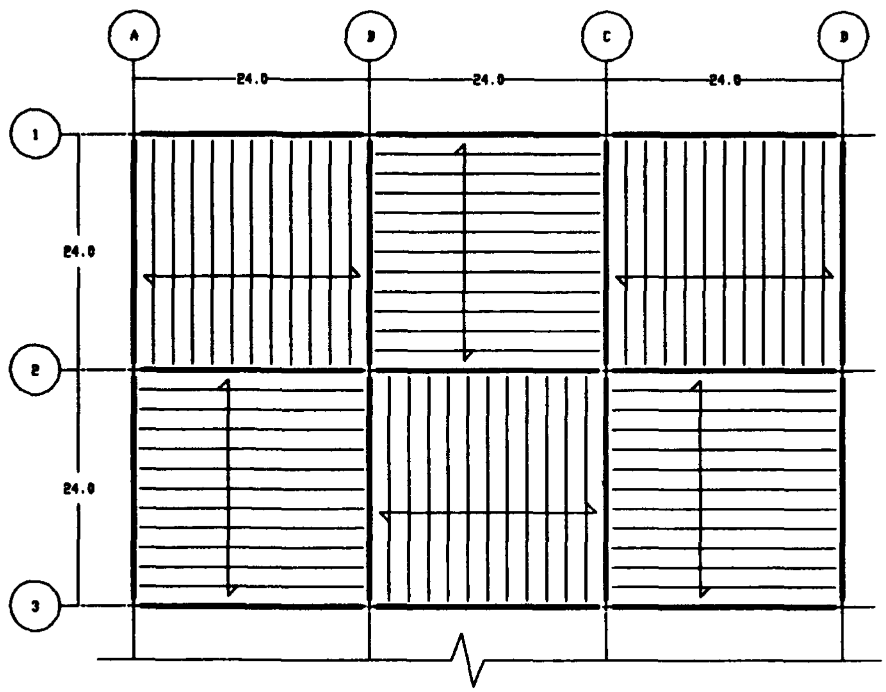
Required: Perform a preliminary analysis and design for the following structural framing options:

1. Open-web steel joists with steel beams on the column lines.

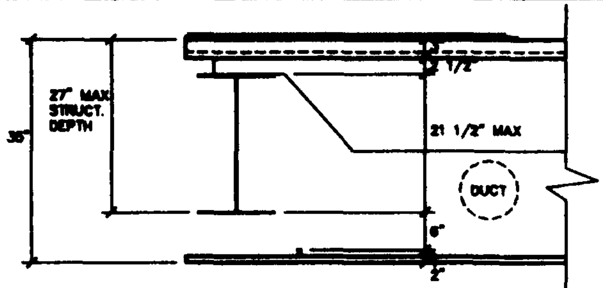
a. All joists spanning in the same direction.

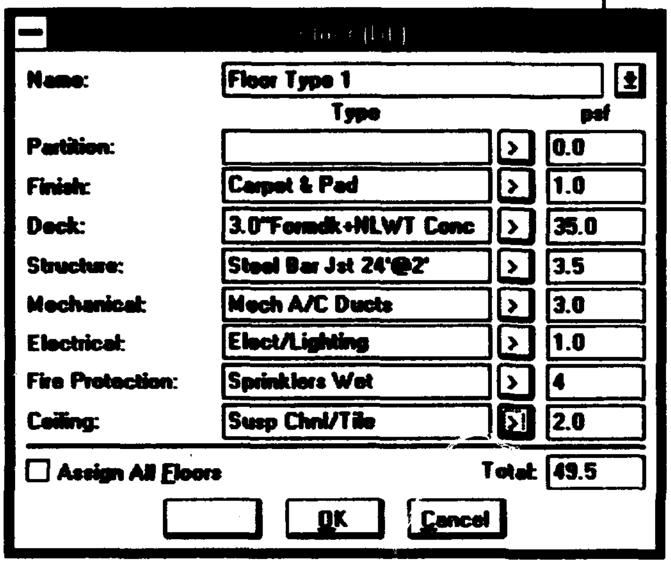


b. Checkerboard arrangement of joists.



Assume the following structural cross section and floor dead loads:





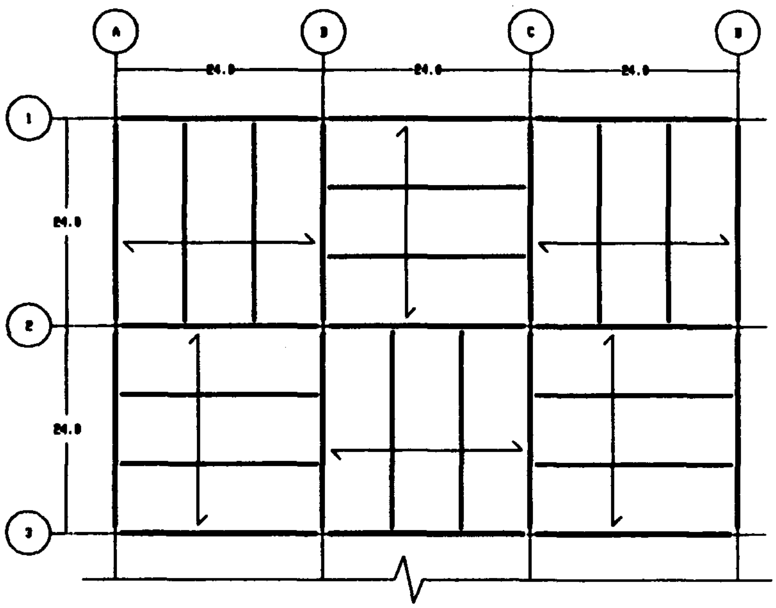
Name: Floor Type 1

	Type	psf
Partition:		0.0
Finish:	Carpet & Pad	1.0
Deck:	3.0"Formdk+NLWT Conc	35.0
Structure:	Steel Bar Jst 24'@2'	3.5
Mechanical:	Mech A/C Ducts	3.0
Electrical:	Elect/Lighting	1.0
Fire Protection:	Sprinklers Wet	4
Ceiling:	Susp Chnl/Tile	2.0

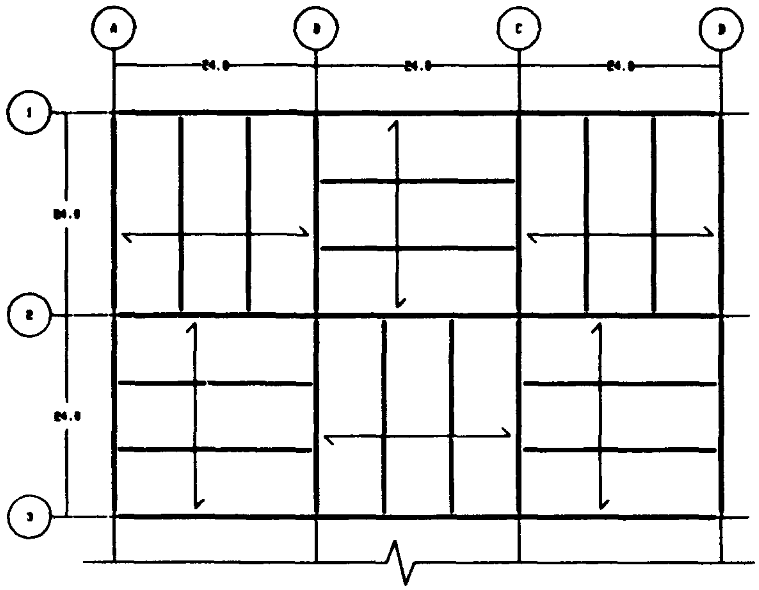
☐ Assign All Floors — Total: 49.5

OK Cancel

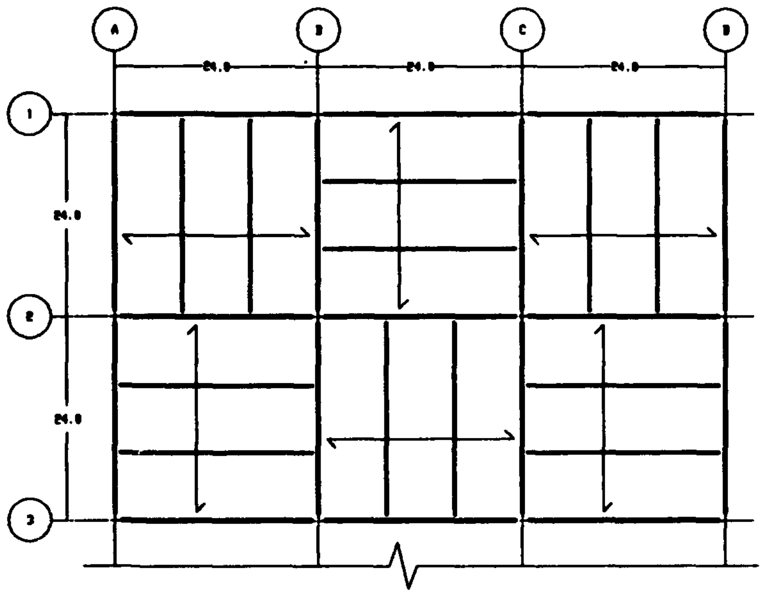
2. Steel beam framing at third points and on the column lines.
   a. Noncomposite construction with checkerboard layout. All connections are simple shear type.



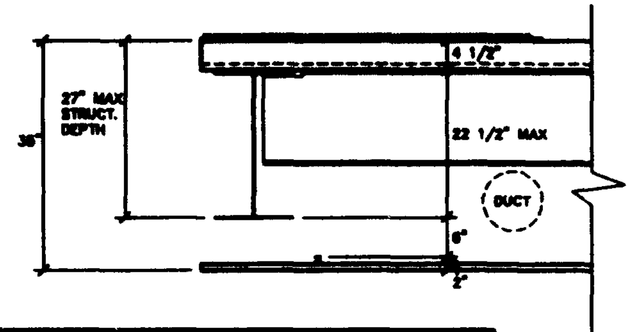
b. Noncomposite construction with checkerboard layout. The girder lines in the short direction of the building are continuous.

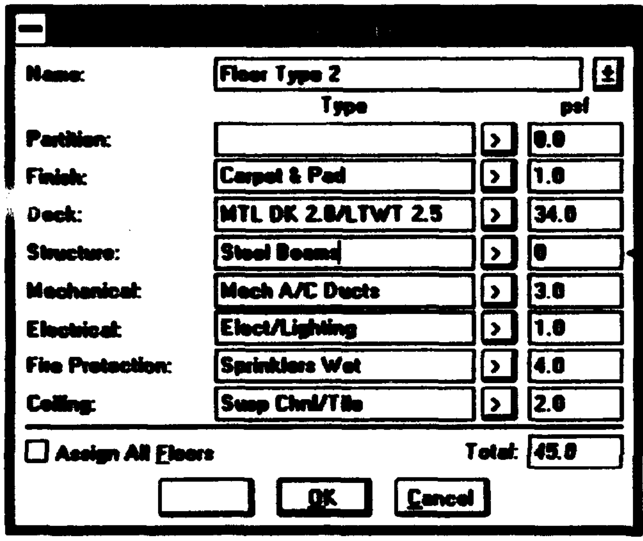


c. Composite construction with checkerboard layout. All connections are simple shear type.



Assume the following structural cross section and floor dead loads:



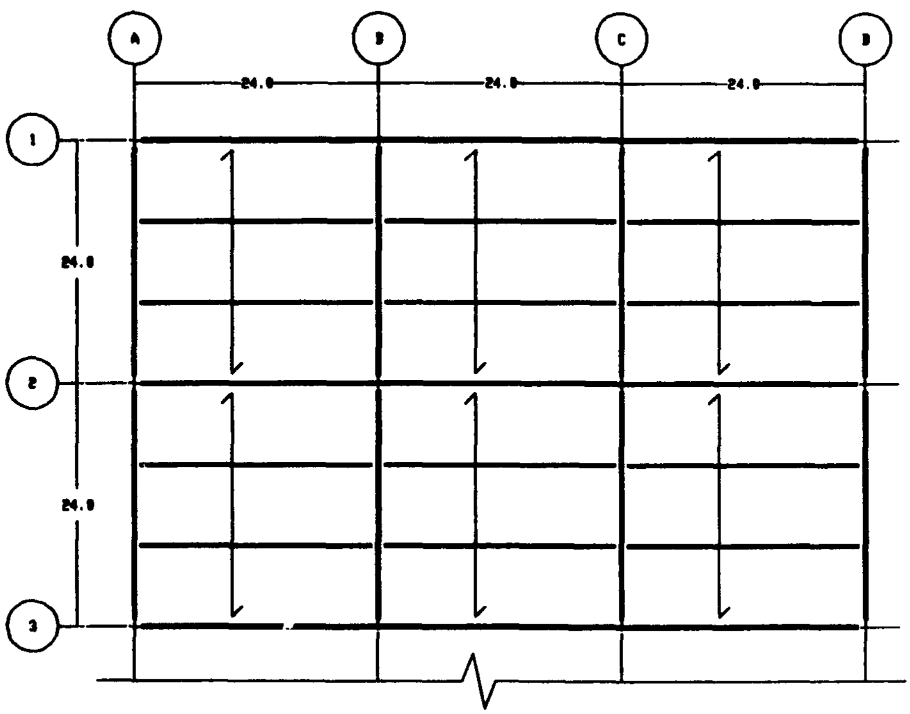


	Type		psf
Name:	Floor Type 2		
Partition:		>	0.0
Finish:	Carpet & Pad	>	1.0
Deck:	MTL DK 2.0/LTWT 2.5	>	34.0
Structure:	Steel Beams	>	0
Mechanical:	Mech A/C Ducts	>	3.0
Electrical:	Elect/Lighting	>	1.0
Fire Protection:	Sprinklers Wet	>	4.0
Ceiling:	Susp Chnl/Tile	>	2.0
Assign All Floors		Total:	45.0

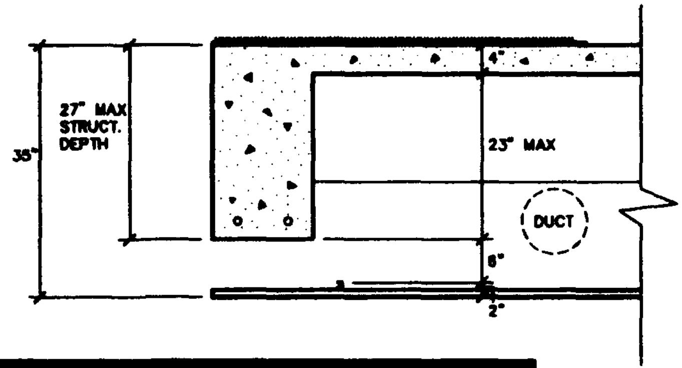
OK Cancel

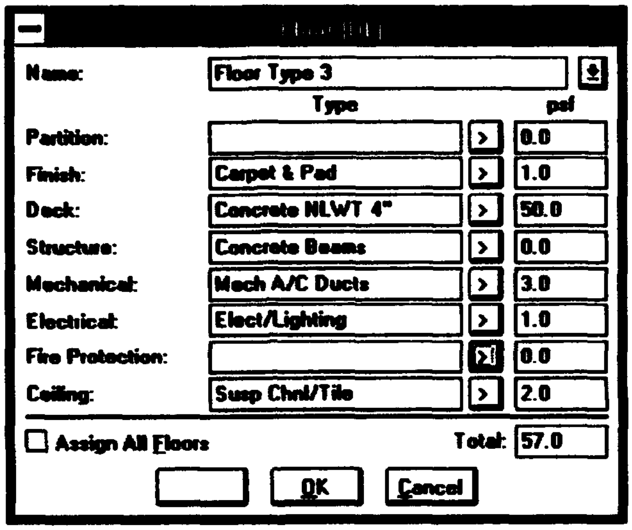
Not part of area load. An estimated beam weight will be provided during Preliminary Analysis.

3. Cast-in-place concrete one-way beam/slab system.



Assume the following structural cross section and floor dead loads:





Name: Floor Type 3

	Type	psf
Partition:		0.0
Finish:	Carpet & Pad	1.0
Deck:	Concrete NLWT 4"	50.0
Structure:	Concrete Beams	0.0
Mechanical:	Mech A/C Ducts	3.0
Electrical:	Elect/Lighting	1.0
Fire Protection:		0.0
Ceiling:	Susp Chnl/Tile	2.0

Assign All Floors — Total: 57.0

OK — Cancel

A. Establish Criteria.

1. Select PROJECT and input the following data:

Project Name	: CORPS OFFICE BUILDING
City/Installation	: VICKSBURG
State	: MS

» Note: This is the only information required, since we will not need snow or wind loads to design floor framing.

B. Draw Volumetric Model.

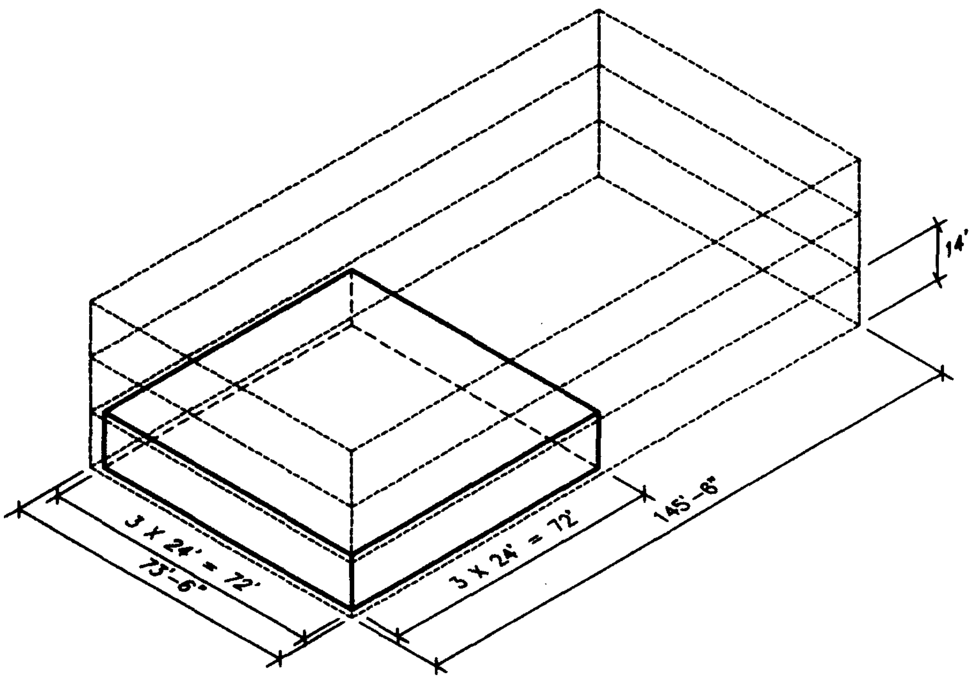
» Note: It is only necessary to draw a portion of the building to study typical bay framing. A one-level, three-bay by three-bay model will be sufficient to design typical interior, exterior, and corner bay members. Consideration of the 9-inch perimeter offset is also not required when studying typical bay framing.

1. Prepare the following basic model on the ground plane:

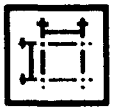
72 feet x 72 feet x 14 feet high.

» Note: It is best not to draw the individual bays in any model. The bays are defined by the structural grid.

» Note: This minimal model will not produce accurate wind load values. The complete building volume, including offsets, is necessary to properly address B/L and h/L factors, as well as the number of floor levels. Snow load values will be correct if no projections are anticipated above the flat roof plane.

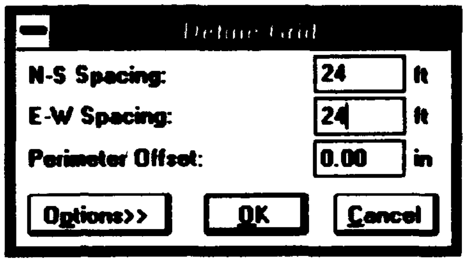


C. Establish the Structural Grid.



>> Note: It is necessary to have a structural grid within which structure can be drawn.

1. Select the DRAW STRUCTURE tool palette.
2. Select DEFINE GRID from the Grid/Opening pull-down menu. A Define Grid dialog window will appear.



3. Set the N-S and E-W SPACING to 24.0 feet.

>> Note: The Perimeter Offset will be left at 0.0 inch as mentioned above. It does not influence the analysis or design of floor framing components.

4. Click on OK when satisfied and the grid will appear on the 3-D model.

D. Define Structurally Significant Openings.

1. Typical floor framing does not include areas where openings occur. None will be inserted for this example.

>> Note: Significant openings would include stairs, elevators, mechanical chases, atriums, skylights, etc.

E. Draw Structure.

>> Note: It is necessary to be in a 2-D view of a plane taken from the 3-D model to draw Structure.

1. Select HORIZONTAL STRUCTURAL PLANE from the View pull-down menu located within the Viewpoint window. Handles will appear on the horizontal planes within the 3-D model for selection.

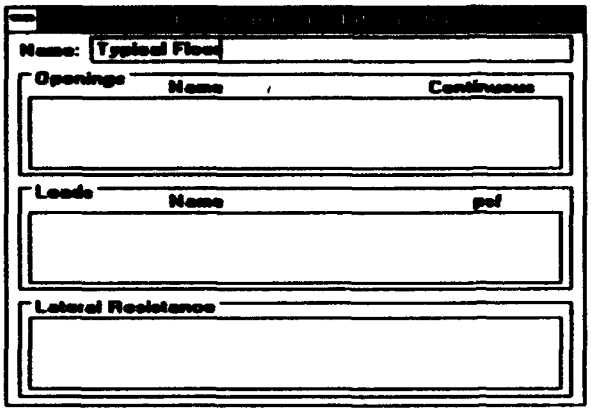
>> Note: CASM defines a roof plane as one which has no objects stacked on it; otherwise, it is a floor plane. Therefore, the top horizontal plane in our model is considered a roof plane, and the plane sitting on the ground plane is considered a floor plane.

>> Note: Planes which are partial roof and partial floor are considered as a floor plane.

2. Select the floor plane (lower handle) from the 3-D model. A 2-D view of the selected plane will appear with the defined structural grid. A North arrow will appear in the lower right hand corner to aid the user in compass orientation. The plane name will also appear in the lower right hand corner.
3. Name the floor plane.

» **Note: All structural planes are automatically uniquely named and numbered.**

a. Select STRUCTURAL PLANE INFORMATION from the View pull-down menu. A Structural Plane Information dialog window will appear showing the plane's name as well as any loads, openings, or lateral resistance locations that have been assigned to it.



b. Change the name to: TYPICAL FLOOR.

c. Select CLOSE from the System pull-down menu in the Structural Plane Information dialog window to remove the window from the screen.

» **Note: An optional way to close the Structural Plane Information dialog window is to reselect Structural Plane Information from the View pull-down menu or double click on the System menu icon.**

» **Note: The Structural Plane Information dialog window will remain on the screen until it is closed.**

4. Draw the narrowly spaced elements (joists).

» **Note: CASM defines narrowly spaced elements as elements that are spaced less than or equal to 4 feet apart and produce distributed reactions on other elements. Widely spaced elements are spaced greater than 4 feet apart and produce concentrated reactions on other elements. Narrowly spaced elements spaced greater than 4 feet produce concentrated reactions on other elements.**

» **Note: There is no need to consider material choice at this time, only the skeletal structural arrangement.**

a. Select NARROWLY SPACED from the Surface/Linear pull-down menu. Handles will appear at the mid points of the grid intersections.

b. Draw joists in the bay defined by grids B to C and 1 to 2. Select a sufficient number of handles in a clockwise order to define the perimeter within which structure is to be drawn. A highlighted dotted line will be drawn showing the perimeter.

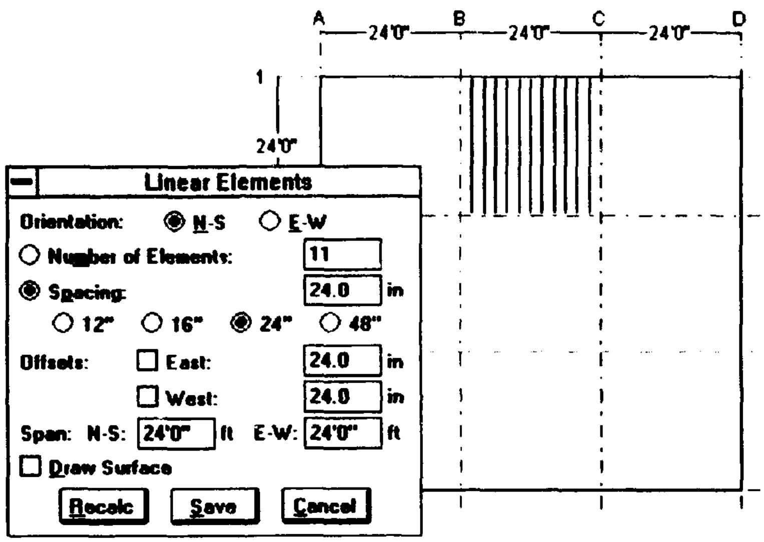
» **Note: A handle represents the midpoint of a line which contains two points on one edge of the perimeter. The perimeter is made up of these points.**

» **Note: There are several ways to select the desired perimeter: (1) by selecting handles in a clockwise order around the entire perimeter, or (2) by selecting one handle and the mutually opposite side handle.**

» **Note: If you select an incorrect handle, double click the right mouse key to stop adding to the perimeter and select CANCEL in the element dialog box.**

c. When the desired perimeter is selected, double click the right mouse key to fix the perimeter. The Linear Elements dialog window appears and narrowly spaced linear elements appear on the screen.

» **Note: The spans indicated in the dialog window are calculated from the selected perimeter.**



d. Revise data in the dialog window to draw joists spaced at 2 feet on center.

(1) Set ORIENTATION to N-S.

(2) Fix the SPACING and select 24 inches.

(3) Turn off DRAW SURFACE.

» **Note: Linear elements are placed by fixing the spacing or fixing the number of elements.**

» **Note: A checkmark in front of a variable fixes that variable.**

» **Note: If neither Offset is fixed, the elements are centered within the perimeter.**

» **Note: The surface element could have been drawn at this time. But, in order to show how to use the surface command, it will be drawn later.**

(4) Click on RECALC to redraw the joists at the new settings. The Number of Elements will be calculated as 11 and the Offsets will be 24.0 inches.

(5) Click on SAVE to fix the joists in the bay.

e. Repeat steps a through d to draw joists in the bay defined by grids B to C and 2 to 3.

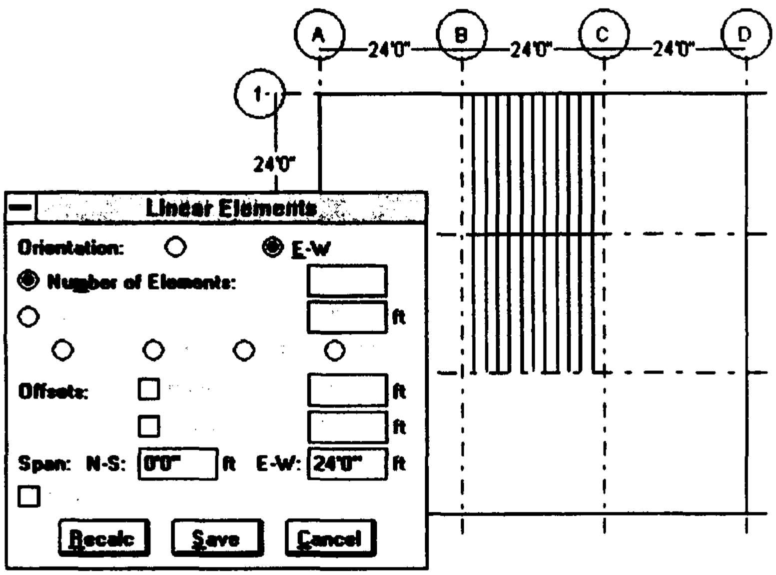
» **Note: Design of a typical girder requires drawing all elements that produce reactions on the girder so that load transfer is complete.**

5. Draw the widely spaced elements (girders).

   a. Select WIDELY SPACED from the Surface/Linear pull-down menu. Handles will appear at the midpoints of all grid lines.

   b. Select the handle between grid intersections B-2 and C-2. The line between those two grid intersections will be highlighted.

   c. Double click the right mouse key to end selection of the handles and add the widely spaced element along the grid line. A Linear Elements dialog window will appear with information about the element. A girder will be drawn.



» **Note: Values in the boxes reflect that only one element has been drawn.**

   d. Select SAVE to fix the girder location.

   e. Repeat steps a through d to add the spandrel girder at grid intersections B-1 and C-1.

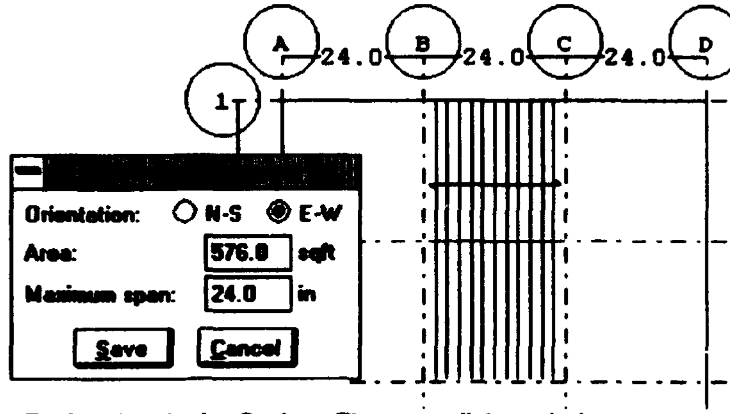
6. Draw the surface elements (decking and slab).

   a. Select ONE-WAY from the Surface/Linear pull-down menu. Handles will appear at the midpoint of the grid lines.

   b. Draw decking and slab in the bay defined by gridlines B to C and 1 to 2. Select a sufficient number of handles in a clockwise order to define the perimeter within which structure is to be drawn. A highlighted dotted line will be drawn showing the perimeter.

   c. Double click the right mouse key to stop defining the perimeter. A Surface Elements dialog window and a one-way surface symbol will appear. The

area of the selected perimeter and the maximum span of the slab/deck are shown.



d. Revise data in the Surface Elements dialog window.

(1) Set ORIENTATION to: E-W.

(2) Select SAVE to add the slab/deck in the bay.

» **Note: It is not necessary to draw the slab/deck in the other bay since narrowly spaced elements are assumed to have a surface above them to distribute loads.**

7. Draw the column and wall structural elements.

» **Note: It is not necessary to draw columns or bearing walls, as support is assumed at the ends of elements.**

8. Manipulation of the structural and geometric model.

» **It is possible to view the structure in 3-D, turn on and off the structure and structural grid with the SHOW STRUCTURE command, print screen, etc.**

F. **Establish independent load cases and assign to the floor plane.**

1. Select the LOADS AND DESIGN tool palette.
2. Prepare floor dead load and name as : FLOOR TYPE 1.

Partition	:None
Finish	:Carpet & Pad
Deck	:3.0" Form deck + NWT Conc.
Structure	:Steel Bar Joist 24" @ 2'
Mechanical	:3.0 psf
Electrical	:1.0 psf
Fire Protection	:Sprinklers Wet 4.0 psf
Ceiling	:Suspended Channel/Tile

The total dead load should be 49.5 psf.

3. Assign the dead load to the floor plane.

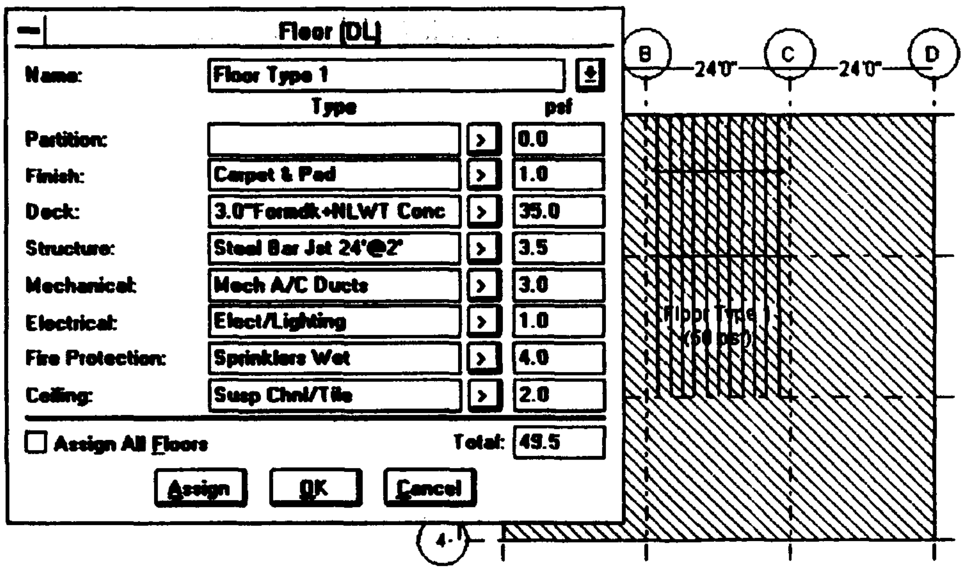
» **Note: You must be in a 2-D view of a selected horizontal or inclined plane to assign loads. You cannot assign dead load or live load to a vertical plane.**

a. Select ASSIGN and a Tributary Area dialog window will appear.

» **The displayed load case is automatically saved when ASSIGN is selected.**

b. Move the mouse pointer to the lower left corner of the plan and click the left mouse key to fix the starting corner of the area to be selected.

c. Move the mouse to the upper right corner of the floor plan and single click the left mouse key again. A hatched texture will appear within the selected area. Its color corresponds to the range within which the magnitude falls. The load name and magnitude will also appear within the selected area.



» **Note: The color of assigned loads will correspond with the following load ranges:**

Blue	: 0 to 59.9 psf
Cyan	: 60.0 to 99.9 psf
Yellow	: 100.0 to 199.9 psf
Red	: 200.0 psf and above

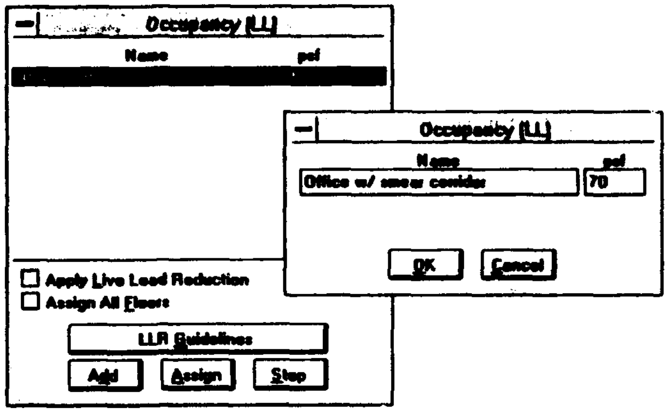
» **Note: The assigned load is automatically saved.**

» **Note: Changing the magnitude of a load will automatically update all the areas to which that load was assigned.**

4. Select OK to end working with the floor dead load.
5. Prepare the Floor live load of offices w/ smear corridor 70 psf.

» **Note: It is necessary to edit the Office Occupancy live load of 50 psf to account for the smeared corridor and partition load. Double click**

anywhere along the Office name/psf line to edit the load magnitude and name as required.



» **Note: We will not use live load reductions in this example due to the smear corridor and partition load inclusion.**

6. Assign the live load to the entire floor plane similar to that done in step 3.

» **Note: Selection of ASSIGN with the mouse pointer will turn off the display of all other loads and turn on the display of the load to be assigned.**

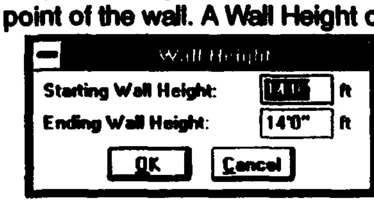
» **Note: The hatched color will be cyan in this case.**

7. Select STOP to end working with the occupancy live load.
8. Prepare the exterior wall dead load and name as: EXTERIOR WALL TYPE 1. The total wall surface load is 73.2 psf.

Finish	:Brick Veneer
Sheathing	:None
Structure	:CMU LT 8"
Insulation	:Exp Polysty Rigid 1"
Finish	:None

» **Note: Wall surface loads are automatically multiplied by the wall height, when assigned, to obtain the linear wall load in plf.**

9. Select ASSIGN and a Tributary Area dialog window will appear.
   a. Move the mouse pointer to grid location A-1 and click the left mouse key to fix the start point of the wall.
   b. Move the mouse pointer to grid location D-1 and click the left mouse key to fix the end point of the wall. A Wall Height dialog window will appear.



» **Note: Walls can only be placed parallel to the N-S or E-W directions. Diagonal walls can be placed by rotating the ground plane.**

c. The default floor to floor height of 14 feet will be used for this example.

d. Click on OK and the wall will be displayed.

>> **Note: The cyan color displayed is based on the psf load value, not the plf value.**

>> **Note: All the dead loads, whether point, linear, or distributed, will be displayed simultaneously, since we are working with dead loads.**

10. Select OK to end work with the wall dead load.

11. This completes the Assigning of loads for the typical deck, joists, and girders.

G. Establish element parameters necessary to design a typical steel open-web joist.

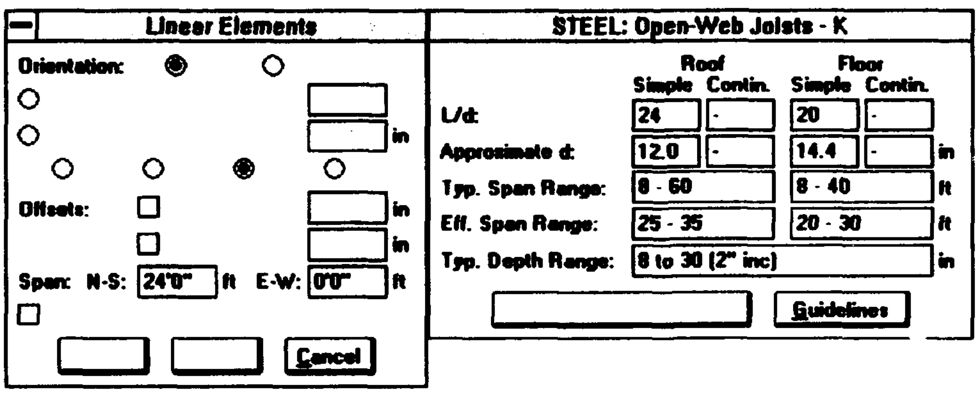
1. Select STEEL from the Materials (mat'l) pull-down menu.

>> **Note: It is necessary to select the material before any other element parameter, since the element type is material dependent.**

>> **Note: A checkmark appears in front of the current material in the pull-down menu and the material icon will be highlighted.**

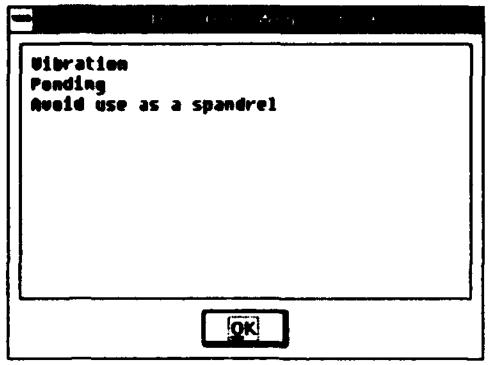
2. Select OPEN-WEB JOISTS-K from the Surface/Linear system category pull-down menu in the Loads and Design tool palette. Handles will appear on all the narrowly spaced elements.

3. Select any handle and click the left mouse key. The selected element will be highlighted by a yellow dashed line. The Linear Elements dialog window will appear showing the dimensions of the selected element. An additional window will appear showing the element attributes. These attributes include span/depth ratio, approximate depth, typical span range, efficient span range, and typical depth range.



4. Review the data shown and select GUIDELINES to be prompted with additional considerations for the element type selected.

>> **Note: You can add other information for an element type by editing the GUIDES.CRD file with the Cardfile program. Refer to the CASM Reference Manual for details.**

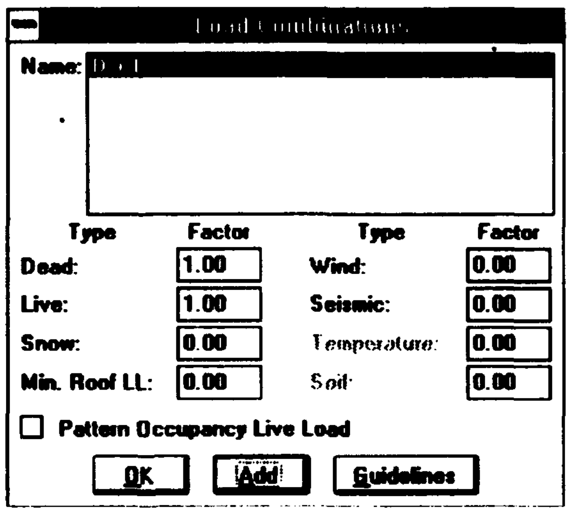


5. Three options exist at this point:
   a. Select CANCEL to end consideration of that element type.
   b. Select a different narrowly spaced element type from the Surf/Line pull-down menu.
   c. Continue on to preliminary analysis with the present element type selected.

H. Preliminary analysis of a typical steel open-web joist.

1. Establish load combination for analysis.

   a. Select LOAD COMBINATIONS from the Loads pull-down menu. The Load Combinations dialog window will appear with a list of the independent loads that can be combined.



›› **Note: It is necessary to set a factor to a value other than zero to include that load in the load combination.**

   b. Select GUIDELINES to review code recommended load combinations for allowable stress design and strength design methods, as well as symbol definitions. Click on OK to erase each overlapping window.
   c. Enter a FACTOR of 1.0 next to the Dead and Live load types.
   d. Select ADD to enter the load combination into the list.

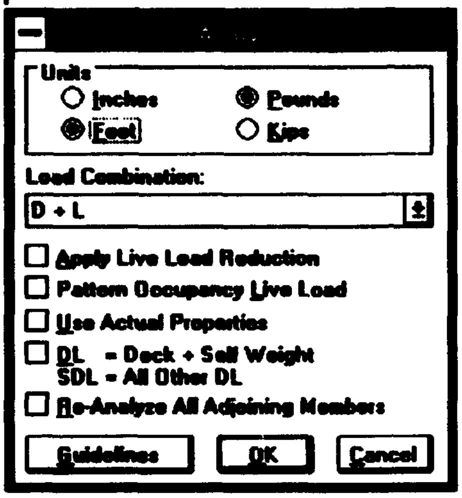
» **Note: The highlighted load combination in the list will be used for the structural analysis. If several load combinations are listed, scroll to the desired load combination.**

e. Do not turn on Pattern Occupancy Live Load for this example.

» **Note: Patterned live load is only of interest for continuous member analysis.**

f. Select OK to end selection of load combinations.

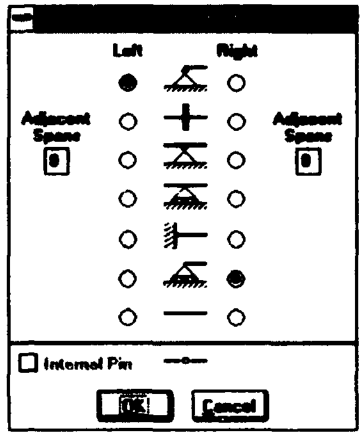
2. Select PRELIMINARY from the Design pull-down menu. An Analysis dialog window will appear.



» **Note: You must have an element selected and a load combination selected to perform a preliminary analysis.**

3. Select appropriate options within the Analysis dialog window.
   a. Select UNITS of Feet and Pounds.
   b. Verify Load Combination of D+ L is selected.
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Do not check DL = DECK + SELF WEIGHT, since the joist is a noncomposite element.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
   h. Select GUIDELINES for information on when to turn on DL = Deck + Self Weight.
   i. Select OK to continue preparation for analysis of the joist element. A Connectivity dialog window will appear, and the left and right ends of the selected element will be highlighted on the floor plane.

» **Note: Selection of CANCEL in any of the Preliminary Analysis dialog windows will stop the process.**



» **Note: Single-screen users can at any time zoom and pan the floor plan as necessary to view the selected element in the most convenient location on the display.**

4. Select appropriate connectivity options.

   a. Create a simple joist span by setting the left support as a <u>HINGE</u> and the right support as a <u>ROLLER</u>.

» **Note: The default connectivity is for a simple span.**

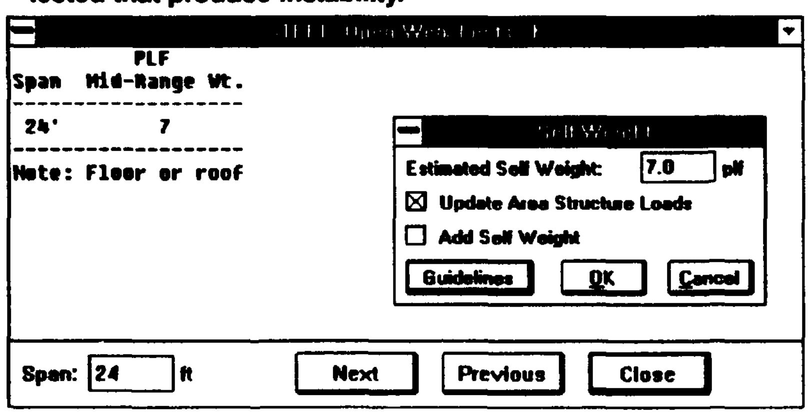
» **Note: The nondefaulted hinge and roller options are used for elements that are continuous over the support.**

   b. Continuous spanning elements require setting the number of adjacent spans to the left or right of the single span selected for analysis. Additional Connectivity dialog windows will appear to select the adjacent span support conditions.

» **Note: The number of adjacent spans to either side cannot be set unless a continuous support type has been selected at that end.**

   c. An internal pin option is available for continuous span conditions and for single span support conditions with sufficient redundancy. An Internal Pin dialog window will appear to set the location of each internal pin.

» **Note: Combinations of left and right end connectivity cannot be selected that produce instability.**

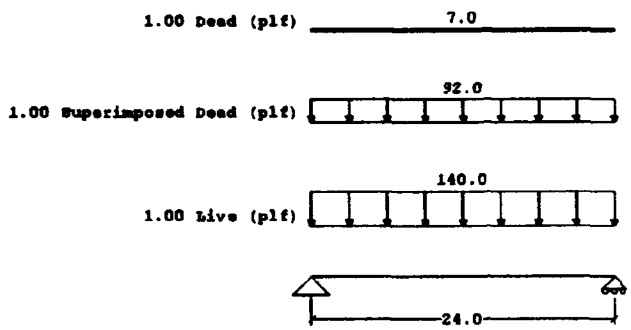


d. Select OK to continue with the preliminary analysis. The tributary area for load calculations on the element will briefly appear on the floor plan followed by the loads and connectivity diagram for the selected load combination for analysis. A Self Weight dialog window will also appear.

5. Select appropriate self weight options.

a. Perform option 2 of the following available options for insertion of the element's self weight:

(1) Use the smeared element self weight called "Structure" in floor dead load type 1. This is an appropriate choice for joists, not the appropriate choice for girders. The plf self weight value is already shown on the Dead load diagram. Leave the estimated self weight value in the dialog window as 0.0 plf.



(2) Insert a new estimated plf self weight. This is an appropriate choice for joists, not the appropriate choice for girders.

(a) Select GUIDELINES. A help dialog window for open-web steel joists will appear with an estimated midrange weight for the joist span.

(b) Estimate the element's self weight with guidance from the weights shown in the help window and type the magnitude in the plf box.

(c) Click on CLOSE to erase the help dialog window from the display.

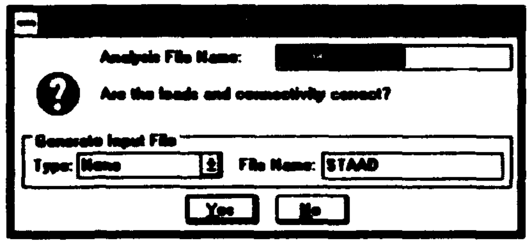
(d) Turn on UPDATE AREA STRUCTURE LOADS if you wish the smeared element self weight, contained in the floor dead load types, to be replaced with the new estimated value. The plf value will be automatically converted to a psf value and its name will be changed to : "est. member weight." The new name will make it easy to recognize that the value was changed.

(3) Add the estimated self weight to the smeared structural dead load. This is the appropriate choice for beams and girders, but not for joists.

(a) Follow steps (a) to (c) of option 2.

(b) Turn on ADD SELF WEIGHT.

b. Click on OK and an Analysis File Name window will appear.



c. It is possible to obtain a hardcopy of the plf load diagrams by selecting PRINT SCREEN from the File pull-down menu.

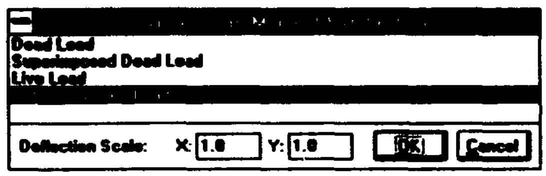
6. Enter the desired ANALYSIS FILE NAME and select YES if the Loads and Connectivity are correct as displayed.

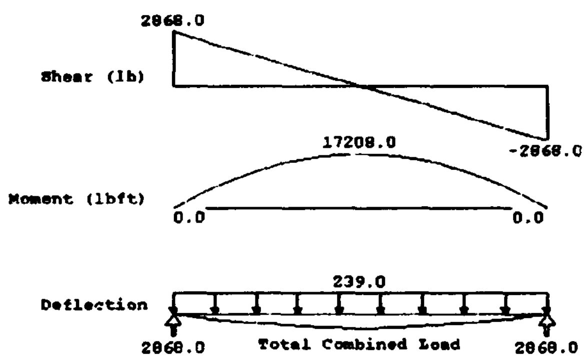
>> **Note: The speed of the analysis can be increased by not assigning a file name; however, no analysis output will be generated for later review.**

>> **Note: Selection of NO ends the preliminary analysis process.**

7. Preliminary analysis of the element begins.

a. When analysis is complete the shear, moment and relative deflection diagrams for the selected load combination will appear on the display. Simultaneously, a View Shear, Moment & Deflection dialog window will appear.





b. Select any of the loads listed in the dialog window to have its shear, moment, relative deflection, loads and reactions displayed.

>> **Note: Analysis is based on default values of E & I = 1.0. Real deflection values are obtained in spreadsheets after E & I are set.**

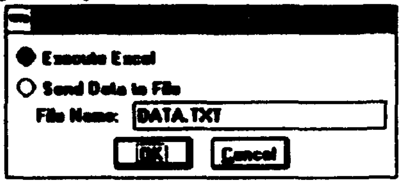
c. It is possible to obtain a hardcopy of the diagrams for any of the load types by selecting PRINT SCREEN from the File pull-down menu.

d. Click on OK to continue to the preliminary design of the joist.

I. Preliminary design of a typical open-web steel joist.

1. Make appropriate selections within the Excel Data dialog window.

   a. The following two options are available:



      (1) Select EXECUTE EXCEL to go directly to the Excel open-web steel joist spreadsheet.

      (2) Select SEND DATA TO FILE and enter a FILE NAME to defer design to a later date.

» **Note: This is the proper choice if you do not have Excel installed or insufficient memory is available to run both CASM and Excel at the same time. Generally, running Windows in standard or enhanced mode will provide sufficient memory.**

   b. Turn on EXECUTE EXCEL.

   c. Click on OK to continue. The CASM program will become an icon, and Excel will be executed loading the open-web steel joist design spreadsheet.

Barjoist Selection

STEEL BAR JOIST PRELIMINARY SELECTION

Project: Corps Office Building	Date: Oct 01, 1991
Location: Vicksburg	Engr:

CASM Load & Analysis Data:

Method: Analysis

Member ID:

Connection: Hinge (Left), Roller (Right)

Span: 24.0 ft

Spacing: 24.0 in

Depth Limit= [illegible] in. max

Fy= 36.0 ksi

Fb= 24.0 ksi

E = 29,000 ksi

Live Defl= L/360= 0.80 in

Total Defl= L/240= 1.20 in

Load Combination: D + L

LoadType	Factored Moment (ft-lb) Left	Mid	Right	Factored Reaction Left(lb)	Right(lb)
Dead		504		84	84
Sup Dead		6,624		1,104	1,104
Live		10,080		1,680	1,680
Lmin Roof					
Snow					
Wind					
Summary		17,208		2,868	2,868

Moment: (EUL)		Reaction: (EUL)	
Total Ld=	239 plf	Total Ld=	239 plf
Live Ld=	140 plf	Live Ld=	140 plf

Analysis Data from CASM Preliminary Analysis

CASM Joist Selection Table: (joist capacities)

Joist Size	Spacing (in)	Total Ld(plf)	Live Ld(plf)	Mmax (ftlb)	Rmax (lb)	Live Ld Defl(in)	Total Ld Defl(in)	Joist Weight (psf)	Joist Weight (plf)
16K2	24.0	254	170	18,288	3,048	0.66	1.13	2.8	5.5
14K3	24.0	245	141	17,640	2,940	0.80	1.36	3.0	6.0
16K3	24.0	283	169	20,376	3,396	0.59	1.02	3.2	6.3
18K3	24.0	320	242	23,040	3,840	0.47	0.80	3.3	6.6

Preliminary Bar Joist selections

CASM Bar Joist Selection:

Joist Size:	16K2	Span:	24.0 ft	Spacing:	24 in	Total Ld:	254 plf	Live Ld:	170 plf
Wgt(tons):	0.07	Mmax:	18,288	Rmax:	3,048	TL defl:	1.13 in	LL defl:	0.66 in

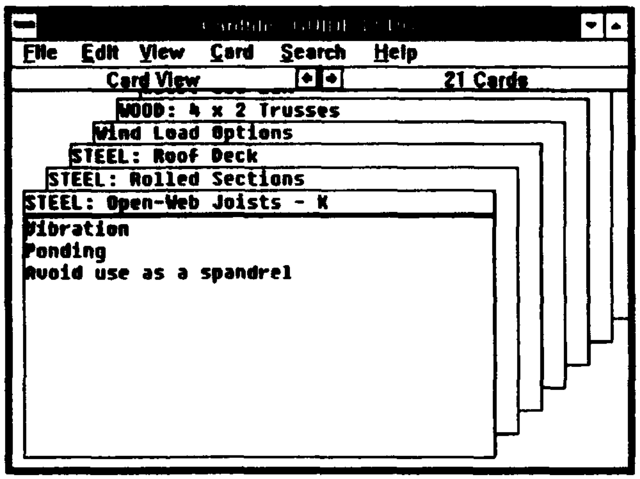
NOTES:

1. Bar joist selections based on 1986 SJI Load Tables.
   Edit spreadsheet stajstk.xls to revise selection table.
2. Approximate moment of inertia of the joist in inches^4 is:
   Ij = 26.767 (WLL) (L^3) (10^-6), where WLL = Live Load value in table;
   where L = Span - 0.33 in feet

» **Note: If there is not enough memory to execute Excel, an error dialog box will appear and you will need to send the data to a file.**

» **Note: If SEND DATA TO FILE was turned on, the data necessary to design the open-web steel joist will be written to the selected file. Proceed directly to step 3.**

2. Design the joist within the Excel design spreadsheet.
   a. The preliminary design spreadsheet and a portion of the CASM joist selection table will appear showing K-Series joist sizes which satisfy the reaction, shear, moment and deflection values for the load combination analyzed, based on criteria of the Steel Joist Institute.
   b. Use the scroll bar to view the remaining portions of the spreadsheet.
   c. Select CARDFILE from the Guidelines pull-down menu to review additional factors which may influence the decision-making process. CLOSE the Cardfile program when finished reviewing the additional factors.



   d. Select the lightest joist size.
      (1) Select SELECT MEMBER from the Member pull-down menu to choose the desired joist size. A Member Size Selection dialog window will appear.
      (2) Click on the 16K2 joist from the list and the selected joist size will be displayed in the lower box within the window.
      (3) If you sent the data directly from CASM, turn on SEND MEMBER SIZE TO CASM to send the joist size and data to CASM.
      (4) Select OK to insert the joist designation in the CASM Bar Joist Selection line in the lower part of the spreadsheet.
   e. Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.
   f. Test the influence of a change in floor finish from carpet to Thin-set terrazzo without going back to CASM.

» Note: Changes to load, span, or spacing cannot be made on the design spreadsheet, since its data are obtained from the analysis done in

CASM. A scratchpad spreadsheet is available to allow varying any of the parameters.

(1) **Select BAR JOIST from the Scratchpad pull-down menu. A steel bar joist scratchpad spreadsheet will appear.**

Barjoist Selection

STEEL BAR JOIST SCRATCH PAD

Project: Corps Office Building	Date: Oct 01, 1991
Location: Vicksburg	Engr:

Load and Analysis Data: (Area Loads)

Method: Analysis — Load Combination: D + L

Member ID:

Connection: Hinge (left), Roller (right)

Span:	24.0 ft
Spacing:	24.0 in
Depth Limit:	28.0 in max
Fy=	36.0 ksi
Fb=	24.0 ksi
E =	29,000 ksi
Live Defl=	L/360= 0.80 in
Total Defl=	L/240= 1.20 in

Load Type	Area Ld (psf)	Load Factors	Factored Load(psf)	Moment (ft-lb)	Reaction (lb)
Dead	3.5	1.00	3.5	504	84
Sup Dead	46.0	1.00	46.0	6,624	1,104
Live	70.0	1.00	70.0	10,080	1,680
Lmin Roof		1.00			
Snow		1.00			
Wind		1.00			
Summary	119.5		119.5	17,208	2,868

Uniform Total Ld= 239 plf
Uniform Live Ld= 140 plf

CASM Joist Selection Table: (joist capacities)

Joist Size	Spacing (in)	Total Ld(plf)	Live Ld(plf)	Mmax (ft-lb)	Rmax (lb)	Live Ld Defl(in)	Total Ld Defl(in)	Joist Weight (psf)	Joist Weight (plf)
16K2	24.0	254	254	18,288	3,048	0.66	1.13	2.8	5.5
14K3	24.0	245	212	17,640	2,940	0.80	1.36	3.0	6.0
16K3	24.0	283	283	20,376	3,396	0.59	1.02	3.2	6.3
18K3	24.0	320	320	23,040	3,840	0.47	0.80	3.3	6.6

Joist Selections (varied spacing, design loads):

Joist Size	Spacing (in)	Total Ld(plf)	Live Ld(plf)	Mmax (ft-lb)	Rmax (lb)	Live Ld Defl(in)	Total Ld Defl(in)	Joist Weight (psf)	Joist Weight (plf)
16K2	24.0	239	140	17,208	2,868	0.66	1.13	2.8	5.5
18K4	36.0	359	210	25,812	4,302	0.60	1.03	2.4	7.2
20K5	48.0	478	280	34,416	5,736	0.57	0.96	2.1	8.2
NONE	60.0	598	350	43,020	7,170				

CASM Bar Joist Selection:

Joist Size:	16K2	Span:	24 ft	Spacing:	24 in	Total Ld:	254 plf	Live Ld:	254 plf
Wgt(tons):	0.07	Mmax:	18,288	Rmax:	3,048	TL defl:	1.13 in	LL defl:	0.66 in

NOTES:

1. Bar joist selections based on 1986 SJI Load Tables.
   Edit spreadsheet stajstk.xls to revise selection table.
2. Approximate moment of inertia of the joist in inches^4 is:
   I = 26.767 (WLL) (L^3) (10^-6), where WLL = Live Load value in table;
   where L = Span - 0.33 in feet

(2) **Use the LOADING/FACTORS selection on the Member pull-down menu of the spreadsheet and change the Superimposed area dead load to 53 psf to reflect a 7-psf dead load increase. Data contained in the scratchpad will be recalculated automatically.**

(3) **Review the changed bar joist sizes and note that the 16K2 still works.**

(4) **Select SELECT MEMBER from the Member pull-down menu to choose the desired joist size. A Member Size Selection dialog window will appear.**

(5) **Click on the 16K2 joist in the list and it will be displayed in the lower box within the window.**

(6) Select OK to insert the joist designation in the CASM Bar Joist Selection line in the lower part of the spreadsheet.

(7) Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.

(8) Select RETURN TO PRELIMINARY from the File pull-down menu to resume work in the Design Spreadsheet.

g. Select RETURN TO CASM from the File pull-down menu. The Excel program will be closed.

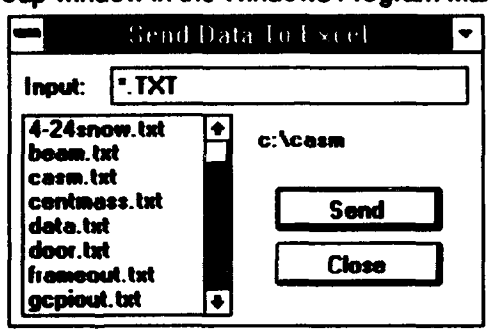
h. Reactivate the CASM program by selecting RESTORE from the System menu.

» **Note: The joist size and spacing is displayed on the joist series selected.**

3. Proceed to the design of another element or revise the preliminary analysis of the current element as desired.
4. Select CANCEL in the Linear Elements dialog window to end work with the selected narrowly spaced element and be able to proceed to the design of the interior rolled shape girder.
5. The following procedure is necessary if SEND DATA TO FILE was selected:

a. SAVE the building project model to a file.

b. Select EXIT from the File pull-down menu to close CASM. Open the CASM group window in the Windows Program Manager.



c. Execute the SENDXL.EXE program to send the data in a file to the Excel spreadsheet programs. A Send Data to Excel dialog window will appear.

d. Select the appropriate datafile name from the list and select SEND. The Send data to Excel program will become an icon and Excel will be executed. The Open-Web Steel Joist Design Spreadsheet will appear.

e. Proceed to step I-2 to select a joist size.

f. RESTORE the Send Data to Excel program from an icon after completion of step I-2g.

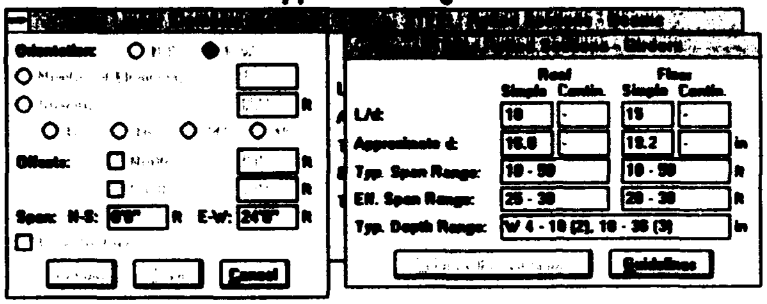
g. Select another file to send to Excel or CLOSE the program window.

h. Execute CASM to design another element.

J. Establish element parameters necessary to design a typical interior steel girder.

1. Steel should still be the selected material.

2. Select ROLLED SECTIONS from the Surf/Line system category pull-down menu. Handles will appear on all the widely spaced elements.
3. Select the girder on gridline 2 between grids B and C. The selected element will be highlighted by a yellow dashed line. The Linear Elements dialog window will appear showing the dimensions of the selected element. Additional windows will appear showing the element attributes.



4. Review the data shown and select GUIDELINES to be prompted with additional considerations for the element type selected.

>> Note: Two element attributes dialog windows exist for steel rolled shapes: one for beams and the other for girders. Click on the visible portion of the hidden dialog window to view the hidden window.

5. Three options exist at this point:
   a. Select CANCEL to end consideration of that element type.
   b. Select a different widely spaced element type from the Surf/Line pull-down menu.
   c. Continue on to preliminary analysis with the present element type selected.

K. Preliminary analysis of a typical interior girder.

1. Select PRELIMINARY from the Design pull-down menu. An Analysis dialog window will appear.

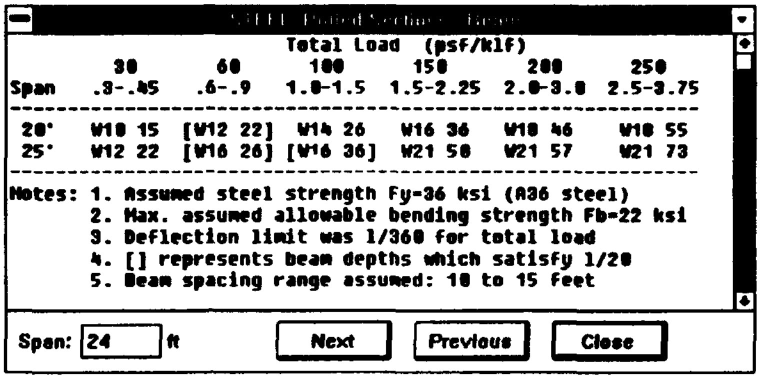
2. Select appropriate options within the Analysis dialog window.
   a. Select UNITS of Feet and Kips.
   b. Verify Load Combination of D+ L is selected.
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Do not check DL = DECK + SELF WEIGHT, since the girder is a noncomposite element.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
   h. Select OK to continue preparation for analysis of the girder element. A Connectivity dialog window will appear, and the left and right ends of the selected element will be highlighted on the floor plane.
3. Select appropriate connectivity options.
   a. Create a simple girder span by setting the left support as a HINGE and the right support as a ROLLER.

b. Select OK to continue with the preliminary analysis. The tributary area for load calculations on the element will briefly appear on the floor plan followed by the loads and connectivity diagram for the selected load combination for analysis. A Self Weight dialog window will also appear.

4. Select appropriate self weight options.

a. Add the estimated self weight to the smeared structural dead load.

(1) Select GUIDELINES. A help dialog window for Steel: Rolled Sections - Beams will appear with an estimated midrange weight for the girder span.



	Total Load (psf/klf)					
	30	60	100	150	200	250
Span	.3-.45	.6-.9	1.0-1.5	1.5-2.25	2.0-3.0	2.5-3.75
20'	W10 15	[W12 22]	W14 26	W16 36	W18 46	W18 55
25'	W12 22	[W16 26]	[W16 36]	W21 50	W21 57	W21 73

Notes: 1. Assumed steel strength Fy=36 ksi (A36 steel)
2. Max. assumed allowable bending strength Fb=22 ksi
3. Deflection limit was 1/360 for total load
4. [] represents beam depths which satisfy 1/20
5. Beam spacing range assumed: 10 to 15 feet

Span: 24 ft Next Previous Close

(2) Estimate the element's self weight with guidance from the weights shown in the help window and type the magnitude in the plf box.

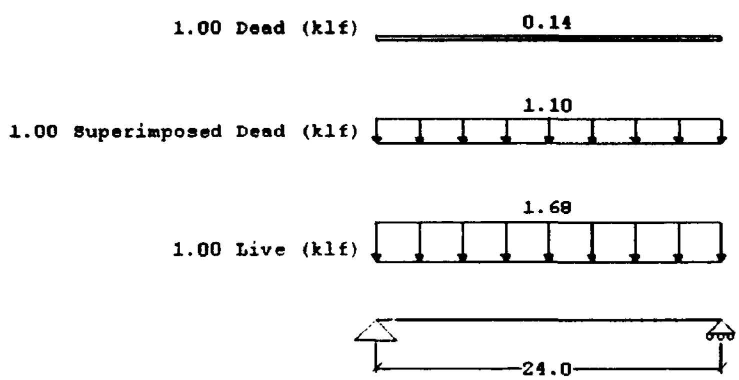
» Note: Add the load values shown on the display to arrive at an approximate total load to find the appropriate load range within the help window. Once a klf total load column is identified, choose an element weight close to the actual span. Estimate the self weight to be 57 plf for this case.

(3) Click on CLOSE to erase the help dialog window from the display.

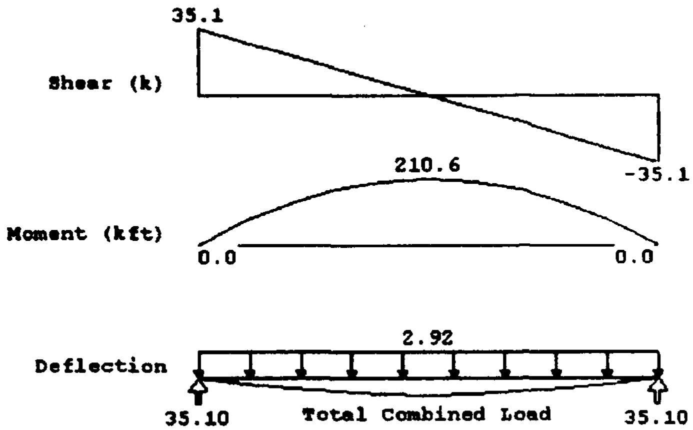
(4) Turn on ADD SELF WEIGHT.

(5) Turn off UPDATE AREA STRUCTURE LOADS.

b. Click on OK and an Analysis File Name window will appear. The klf load shown on the dead load diagram will be updated to include the estimated self weight.



5. Enter the desired ANALYSIS FILE NAME and select YES if the Loads and Connectivity are correct as displayed.
6. Preliminary analysis of the element begins.
   a. When analysis is complete, the shear, moment, relative deflection, loads and reactions diagrams for the selected load combination will appear on the display. Simultaneously, a View Shear, Moment & Deflection dialog window will appear.



   b. Select any of the loads listed in the dialog window to have its shear, moment, relative deflection, loads and reactions displayed.
   c. Click on OK to continue to the preliminary design of the girder.

L. Preliminary design of a typical girder.
1. Make appropriate selections within the Excel Data dialog window.
   a. This time, turn on SEND DATA TO FILE and change the FILE NAME to BEAM1.TXT.
   b. Click on OK to continue. The data necessary to design the steel girder will be written to the selected file.
2. Send the girder data file to Excel outside of CASM.
   a. Select CANCEL in Linear Elements dialog window; then SAVE the building project model to a file called STRUCT1.BLD.
   b. Select EXIT from the File pull-down menu to close CASM. Open the CASM group window in the Windows Program Manager.
   c. Execute the SENDXL.EXE program to send the data in a file to the Excel spreadsheet program. A Send Data to Excel dialog window will appear.
   d. Select the data file name BEAM1.TXT from the list and select SEND. The Send data to Excel program will become an icon and Excel will be executed. The Steel Beam Design Spreadsheet will appear.
3. Design the girder within the Excel design spreadsheet.
   a. The Preliminary Design Spreadsheet will appear, including a portion of the CASM beam selection table which shows wide flange sizes that satisfy the reaction, shear, moment and deflection values for the load combination analyzed.

Steel Beam Selection

STEEL BEAM PRELIMINARY SELECTION

Project: Corps Office Building	Date: Oct 01, 1991
Location: Vicksburg	Engr:

CASM Load & Analysis Data:

Method: Analysis   Load Combination: D + L

Member ID:
Connectivity: Hinge (Left), Roller (Right)
Beam Span: 24.0 ft
Trib Width= 24.0 ft
Depth Limit= 21.5 in. max
Fy= 36.0 ksi
Fb=.66*Fy= 24.0 ksi
Fv= 14.4 ksi
E = 29,000 ksi
Live Ld Defl= L/360 =0.80 in
Total Defl= L/240 =1.20 in

Load Type	Factored Moments (k-ft) Left	Mid	Right	Fact. Reactions Left(k)	Right(k)
Dead		10.2		1.7	1.7
Sup Dead		79.5		13.3	13.3
Live		121.0		20.2	20.2
Lmin Roof					
Snow					
Wind					
Summary		210.6		36.1	36.1

Max: M=	210.6 k-ft	R=	36.1 kips
Sx(req)=	105.3 in^3	Ix(req)=	540.6 in^4

CASM Beam Selection Table:

Beam	Depth d (in)	Width bf (in)	Ix (in^4)	Sx (in^3)	Live Ld Defl (in)	Total Ld Defl (in)	Shear fv (ksi)	Bending fb (ksi)	Beam Wt (lb)
W 12 x 79	12.4	12.08	662	107	-0.65	-1.14	6.0	23.6	1,896
W 18 x 60	18.2	7.56	984	108	-0.44	-0.77	4.8	23.4	1,440
W 21 x 57	21.1	6.56	1,170	111	-0.37	-0.64	4.1	22.8	1,368
W 14 x 74	14.2	10.07	796	112	-0.54	-0.95	5.5	22.6	1,776
W 16 x 67	16.3	10.24	954	117	-0.45	-0.79	5.4	21.6	1,608

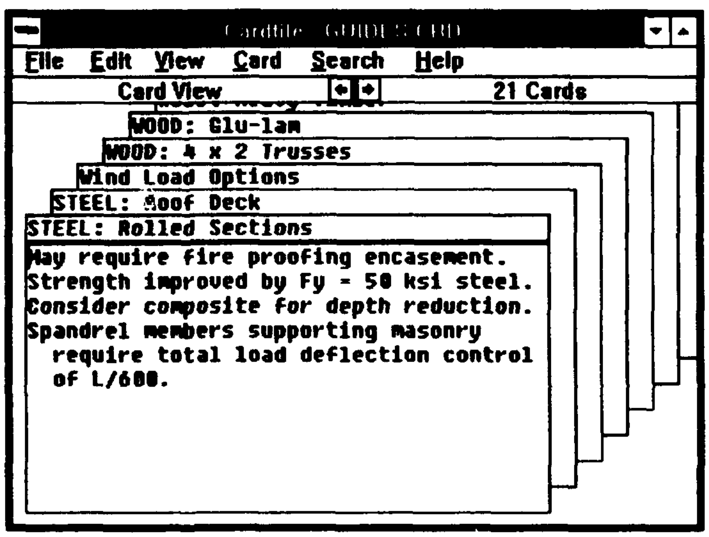
CASM Steel Beam Selection:

						Live / Total	
W 21 x 57	Span= 24.0 ft	Ix=	1,170	Sx=	111	Defl(in): -0.37	-0.64
		fv=	4.1	fb=	3.8	Beam Wt(tons)=	0.68

Notes:
1. Steel beam properties from ASD - AISC Steel Construction Manual, 9th edition

b. Use the scroll bar to view the nonvisible portions of the spreadsheet.

c. Other system factors that may influence your selection can be found in CARDFILE from the Guidelines pull-down menu. CLOSE the Cardfile program when finished reviewing the additional factors.



d. Set the depth limit by using the Member pull-down menu to 21.5 inches and select the lightest steel beam size. Based on the depth limit, the lightest section is a W21x57.

(1) Select SELECT MEMBER from the Member pull-down menu to choose the desired beam size. A Member Size Selection dialog window will appear.

(2) Click on the selected beam in the list and the selected beam size will be displayed in the lower box within the window.

(3) Turn off SEND MEMBER SIZE TO CASM since the data was not sent directly from CASM.

(4) Select OK to insert the beam designation in the CASM Beam Selection line in the lower part of the spreadsheet.

e. Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.

f. Test the influence of a change in floor finish from carpet to Thin-set terrazzo without having to go back to CASM.

» **Note: Changes to load, span or spacing cannot be made on the preliminary design spreadsheet, since its data are obtained from the analysis done in CASM. A scratchpad spreadsheet is available to allow varying any of the parameters.**

(1) Select STEEL WIDE FLANGE from the Scratchpad pull-down menu. A steel beam scratchpad spreadsheet will appear.

Steel Beam Selection

STEEL BEAM SCRATCH PAD

Project: Corps Office Building	Date: Oct 01, 1991
Location: Vicksburg	Engr:

Load & Analysis Data:

Method: Analysis — Load Combination: D + L

		Load Type	Area Ld(psf)	Load Factors	M+(max) (kip-ft)	M-(max) (kip-ft)	R(max) (kip)
Member ID:							
Connectivity:	Hinge (Left)						
	Roller (Right)	Dead	5.9	1.00	10.2		1.7
Bm Span:	24.0 ft	Sup Dead	46.0	1.00	79.5		13.2
Trib Width:	24.0 ft	Live	70.0	1.00	121.0		20.2
Depth Limit:	21.5 in max	Lmin Roof		1.00			
Fy=	36.0 ksi	Snow		1.00			
Fb=.66*Fy=	24 ksi	Wind		1.00			
Fv=	14.4 ksi	Summary	121.9		210.6		35.1
E=	29,000 ksi	Fact. Unif Ld=	2.9 klf				
Live Ld Defl=	L/360 =0.80in	Max: M=	210.6 kip-ft		R=	35.1 kips	
Total Defl=	L/240 =1.20in	Sx(req)=	106.4 in^3		Ix(req)=	626.6 in^4	

Beam Selection Table:

END CONDITIONS: Simple(S); Two Span(D); Continuous(C); Fixed(F)= S

Beam Size	Depth d (in)	Width bf (in)	Ix (in^4)	Sx (in^3)	Live Ld Defl(in)	Total Ld Defl(in)	Shear fv (ksi)	Bending fb (ksi)	Beam Wt(lbs)
W 12 x 79	12.4	12.08	662	107	0.65	1.14	6.0	23.6	1,896
W 18 x 60	18.2	7.56	984	108	0.44	0.76	4.6	23.4	1,440
W 21 x 57	21.1	6.56	1,170	111	0.37	0.64	4.1	22.8	1,368
W 14 x 74	14.2	10.07	796	112	0.54	0.94	5.5	22.6	1,776
W 16 x 67	16.3	10.24	954	117	0.45	0.79	5.4	21.6	1,608

CASM Steel Beam Selection: Live / Total

W 21 x 57	Span= 24.0 ft	Ix= 1,170	Sx= 111	Defl(in): 0.37	0.64
		fv= 4.12	fb= 22.8	Beam Wt(tons)=	0.68

Notes:

1. Steel beam properties from ASD - AISC Steel Construction Manual, 9th edition
2. Moments and shears are based on uniform loads or EUL.
3. The moment factors for the different end conditions are:

End condition	Positive Moment	Negative Moment
Simple (S) :	.125wL^2	0
Two Span (D) :	.1025wL^2	.125wL^2
Continuous (C) :	.1013wL^2	.1167wL^2
Fixed (F) :	.0417wL^2	.0833wL^2

(2) Using the LOADS/FACTORS option on the Member pull-down menu, change the Superimposed area dead load to 53 psf to reflect a 7-psf dead load increase. Data contained in the scratchpad will be recalculated automatically.

(3) Set the depth limitation to 21.5 inches using the Member pull-down menu.

(4) Review the changed beam sizes in the Beam Selection Table and select the lightest section as the W18x65.

(5) Select SELECT MEMBER from the Member pull-down menu to choose a desired beam size. A Member Size Selection dialog window will appear.

(6) Click on the selected beam designation in the list and the selected beam size will be displayed in the lower box within the window.

(7) Select OK to insert the beam designation in the CASM Beam Selection line in the lower part of the spreadsheet.

(8) Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.

(9) Select RETURN TO PRELIMINARY from the File pull-down menu to resume work in the Design Spreadsheet.

g. Select RETURN TO CASM from the File pull-down menu. The Excel program will be closed.

4. Return to the CASM program.

a. CLOSE the SendXL program.

b. Execute CASM and load the building project model STRUCT1.BLD. The last view of the floor plane will be displayed on the screen.

c. You are now able to continue to the design of another element.

CASM

M. Establish element parameters necessary to design a typical spandrel steel girder.

1. Steel should still be the selected material.
2. Select ROLLED SECTIONS from the Surface/Linear system category pull-down menu.
3. Select the spandrel girder on gridline 1 between grids B and C.
4. Review the data shown and select GUIDELINES to be prompted with additional considerations for the element type selected.
5. Continue on to preliminary analysis with the present element type selected.

N. Preliminary analysis of a typical spandrel girder.

» **Note: The exterior wall weight will be included in the superimposed dead load magnitude.**

1. Select PRELIMINARY from the Design pull-down menu.
2. Select appropriate options within the Analysis dialog window.

a. Select UNITS of Feet and Kips.

b. Verify Load Combination of D+ L is selected.

c. Do not check APPLY LIVE LOAD REDUCTION.

d. Do not check PATTERN OCCUPANCY LIVE LOAD.

e. Do not check USE ACTUAL PROPERTIES.

f. Do not check DL = DECK + SELF WEIGHT, since the girder is a noncomposite element.

g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.

h. Select OK to continue preparation for analysis of the girder element.

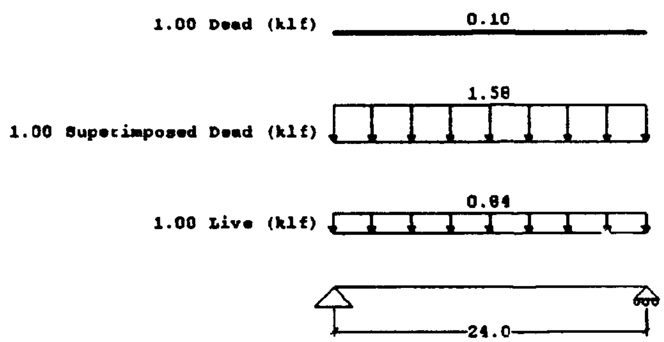
3. Select appropriate connectivity options.

a. Create a simple girder span by setting the left support as a HINGE and the right support as a ROLLER.

b. Select OK to continue with the preliminary analysis.

4. Select appropriate self weight options.

a. Add an estimated self weight of 57 plf to the smeared structural dead load. This magnitude was estimated based on a summation of the loads shown on the display (approximately 2.4 klf) and the span of 24 feet.

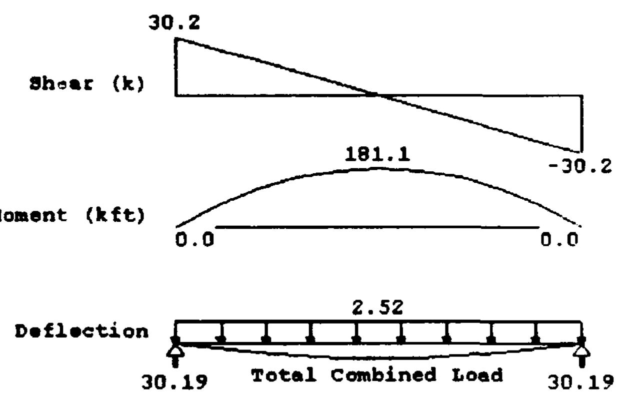


b. Click on OK to continue.

5. Enter the desired ANALYSIS FILE NAME and select YES if the Loads and Connectivity are correct as displayed.

6. Preliminary analysis of the element begins.

a. When analysis is complete the shear, moment and relative deflection diagrams for the selected load combination will appear on the display.



b. Select any of the loads listed in the dialog window to have its shear, moment, relative deflection, loads and reactions displayed.

c. Click on OK to continue to the preliminary design of the spandrel girder.

O. **Preliminary design of a typical spandrel girder.**

1. Make appropriate selections within the Excel Data dialog window.

a. Choose one of the following two options :

(1) Select EXECUTE EXCEL to go directly to the Excel steel beam spreadsheet.

(2) Select SEND DATA TO FILE and enter a FILE NAME to defer design to a later date.

>> **Note: This is the proper choice if you had difficulty in running Excel from CASM. Follow the procedure described in step L-2 to pass the data on to Excel.**

b. Click on OK to continue.

2. Design the spandrel girder within the Excel design spreadsheet.

a. The Preliminary Design Spreadsheet will appear.

b. Use the scroll bar to view the nonvisible portions of the spreadsheet.

c. Use CARDFILE to review factors that may influence your decisions.

>> **Note: Spandrel members supporting brick masonry shall have their total load deflection limited to L/600.**

d. No depth limit is required for the spandrel girder; however, set the TOTAL DEFL to 600. The W24x55 is the lightest steel beam size displayed in the selection table.

(1) Select SELECT MEMBER from the Member pull-down menu to choose the desired beam size.

(2) Click on the selected beam in the list.

(3) Turn on SEND MEMBER SIZE TO CASM if the data was sent directly from CASM.

(4) Select OK to insert the beam designation in the CASM Beam Selection line in the lower part of the spreadsheet.

e. Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.

f. Select RETURN TO CASM from the File pull-down menu. The Excel program will be closed.

g. If you activated Excel through CASM, reactivate the CASM program by selecting MAXIMIZE from the System menu if you are using a dual-screen workstation. Select RESTORE if you are using the single-screen workstation.

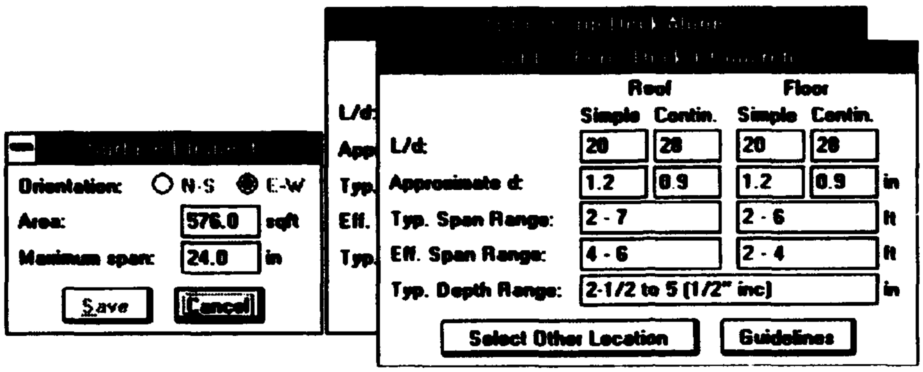
>> **Note: If data were sent to a file, re-execute CASM and load your previous project data file.**

3. Proceed to the design of another element or revise the preliminary analysis of the current element as desired.

4. Select CANCEL in the Linear Elements dialog window to end work with the selected widely spaced element and to be able to proceed to the design of the metal form deck.

P. Establish element parameters necessary to design the metal form deck.

1. Steel should still be the selected material.
2. Select FORM DECK from the Surface 1-Way types contained in the Surf/Line system category pull-down menu. Handles will appear on the one-way surface element symbols.
3. Select the handle within the bay between the grids B to C and 1 to 2.
4. Review the two attribute dialog windows shown and select GUIDELINES to be prompted with additional considerations for the element type selected.



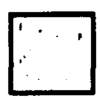
5. Continue on to preliminary analysis.

Q. Preliminary analysis of the metal form deck alone.

1. Change the current load combination for form deck analysis to dead load only.

» Note: The wet concrete and the deck weight are the only loads acting on the form deck.

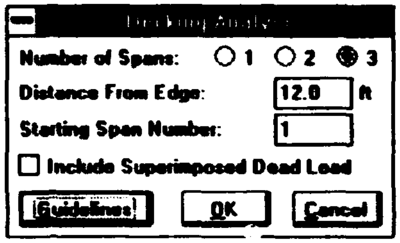
   a. Change the LIVE load FACTOR to 0.0. The dead load factor is currently 1.0.
   b. Select ADD to add the dead only combination to the list and make it current.
   c. Click on OK to end working with load combinations.



2. Select PRELIMINARY from the Design pull-down menu.
3. Select appropriate options within the Analysis dialog window.
   a. Select UNITS of Feet and Pounds.
   b. Verify Load Combination of D is selected.
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Do not check DL = DECK + SELF WEIGHT, since the form deck is a noncomposite element.

g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.

h. Select OK to continue preparation for analysis of the form deck element. The Decking Analysis dialog window will appear instead of the connectivity dialog window. The deck spans will be numbered across the bay, and one edge will be highlighted yellow on the display.



4. Select appropriate decking analysis options.

   a. Select the appropriate number of spans as three.

   b. Locate the position of the typical 1-foot strip for analysis.

      (1) The distance from the highlighted edge to the centroid of the 1-foot strip is defaulted to the center of the bay and is shown in the dialog window as 12.0 feet. This distance can be changed to re-position the 1-foot strip where desired.

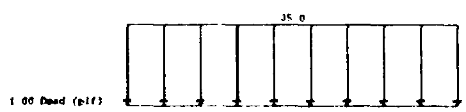
      (2) Select the Starting Span Number of the three spans to be analyzed. This positions the three spans within the bay along the 1-foot strip.

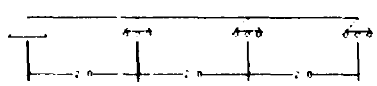
   c. Review GUIDELINES for information on whether to include superimposed dead load.

   d. Turn off INCLUDE SUPERIMPOSED DEAD LOAD.

   e. Select OK to continue with the preliminary analysis.

5. Enter the desired ANALYSIS FILE NAME and select YES if the Loads and Connectivity are correct as displayed.





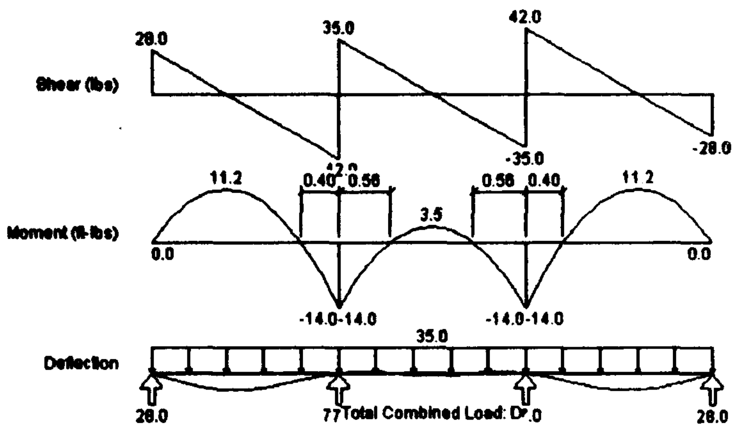
» **Note: Connectivity is automatically set so the first support is a hinge and all remaining supports are rollers. Thus, the deck is continuous over the three spans.**

6. Preliminary analysis of the element begins.

   a. When analysis is complete the shear, moment and relative deflection diagrams for the selected load combination will appear on the display.

b. **Select any of the loads listed in the dialog window to have its shear, moment, relative deflection, loads and reactions displayed.**

» **Note: For this analysis there is only one individual load type.**



c. **Click on OK to continue to the preliminary design of the form deck alone.**

R. Preliminary design of the metal form deck alone.

1. **Select 9/16"-26 GA form deck. Refer to section O.1. through O.4. for use of the Preliminary Design spreadsheet.**

STEEL FORM DECK PRELIMINARY DESIGN

Project: Date: Sep 18 1993
Location: Engr:

Load and Analysis Data:

Method: Analysis Load Combination: D

			Factored Moment (ft-lb)			Factored Reaction	
Member ID:		LoadType	Left	Mid	Right	Left(lb)	Right(lb)
Connectivity:	Beam (Left)						
	Beam (Right)	Dead	14	11	14	42	42
Deck Span:	2.0 ft	Sup Dead					
Trib Width=	12.0 in	Live					
Depth Limit=	1.0 in. max	Lmin Roof					
Fy=	33.0 ksi	Snow					
Fb=	20.0 ksi	Wind					
Fv=	13.2 ksi	Summary	14	11	14	42	42

E = 29,000 ksi		Fire Rating 1.00 Hrs
Live Ld Defl= L/240 =0.10 in	Depth	Min thickness above deck:
Total Defl= L/180 =0.13 in	Load #1: Deck+Slab+20psf	3.50 inches
CASM Slab Design:	Load #2: Deck+Slab+150#	Reqd depth: 4.50 inches
Fy= 60 ksi	Depth= 5.50 in	Reqd As= 0.002 in^2
F'c= 4.0 ksi	d= 2.25 in	Selected WWF: 6x6-W1.4xW1.4
Weight= 145 pcf	Mmax= 14 ft-lb	Mu= 280.9059 ft-lb

Form Deck Code Load Combinations:

	Case	Load (psf)	Fb Factor	M+ (f-lb)	M- (f-lb)	Sx+ (in.3)	Sx- (in.3)	Ix (in.4)
Simple Span	#1	86.5	1.00	43.2		0.026		0.006
1	#2	66.5	1.33	108.2		0.049		0.011
Maximums:				108.2		0.049		0.011

S. Save the scheme 1a project data file as SCHEME1A.BLD.

[illegible]

A. OPEN the scheme 1a datafile name: SCHEME1A.BLD.The steps contained in A through D for scheme 1a are the same.

B. Draw Structure.

1. Select the DRAW STRUCTURE tool palette.

2. Delete the narrowly spaced elements within the center bay, which is bounded by grids B and C between grids 2 and 3.

   a. Select DELETE STRUCTURE from the Edit pull-down menu. Handles will appear on each structural element to delete.

» Note: Only one handle will appear for a series of elements within a bay. It is not possible to delete a single element within a series of elements.

   b. Select the joist handle contained in the center bay and all the joists will be removed from the display.

   c. Double click the right mouse key to exit the delete structure command.

3. Insert narrowly spaced elements spanning in the east/west direction within the center bay spaced at 24 inches on center. Turn on DRAW SURFACE to insert a one-way surface of metal form deck and concrete fill above the joists.

» Note: Since the joists run parallel to the interior girder, which we intend to design, a surface element needs to be drawn across the rectangle of space in between the girder and the adjacent parallel joist so that the rectangle is not interpreted by CASM to be an opening.

C. The independent load cases are the same as for scheme 1A.

1. Select the LOADS AND DESIGN tool palette.

D. Establish element parameters necessary to design a typical interior steel girder.

1. Steel should still be the selected material.

2. Select ROLLED SECTIONS from the Surface/Linear system category pull-down menu.

3. Select the typical interior girder on gridline 2 between grids B and C.

4. Review the data shown and select GUIDELINES to be prompted with additional considerations for the element type selected.

5. Continue on to preliminary analysis with the present element type selected.

E. Preliminary analysis of a typical interior girder.

1. Use the dead + live load combination for analysis.

2. Select PRELIMINARY from the Design pull-down menu.

3. Select appropriate options within the Analysis dialog window.

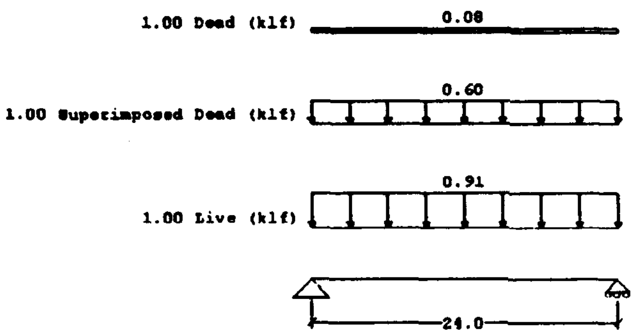
   a. Select UNITS of Feet and Kips.

   b. Verify Load Combination of D+ L is selected.

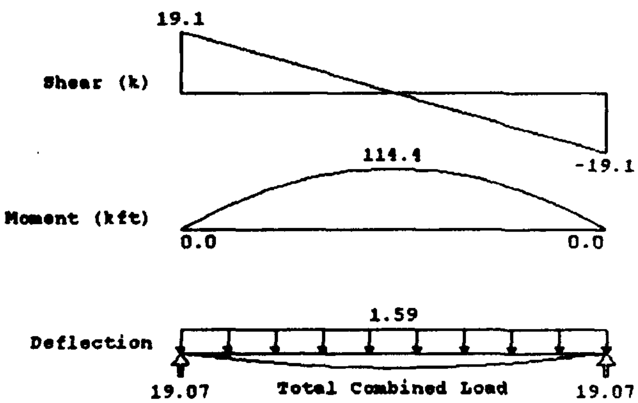
   c. Do not check APPLY LIVE LOAD REDUCTION.

d. Do not check PATTERN OCCUPANCY LIVE LOAD.
e. Do not check USE ACTUAL PROPERTIES.
f. Do not check DL = DECK + SELF WEIGHT, since the girder is a noncomposite element.
g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
h. Select OK to continue preparation for analysis of the girder element.

4. Select appropriate connectivity options.
   a. Create a simple girder span by setting the left support as a HINGE and the right support as a ROLLER.
   b. Select OK to continue with the preliminary analysis.
5. Select appropriate self weight options.
   a. Add an estimated self weight of 36 plf to the smeared structural dead load. This magnitude was estimated based on a summation of the loads shown on the display (approximately 1.56 klf) and the span of 24 feet.



   b. Click on OK to continue.
6. Enter the desired ANALYSIS FILE NAME and select YES if the Loads and Connectivity are correct as displayed.
7. Preliminary analysis of the element begins.
   a. When analysis is complete the shear, moment, relative deflection, loads and reaction diagrams for the selected load combination will appear on the display.



b. Select any of the loads listed in the dialog window to have its shear, moment, relative deflection, loads and reactions displayed.

c. Click on OK to continue to the preliminary design of the typical interior girder.

F. Preliminary design of a typical interior girder.

1. Make appropriate selections within the Excel Data dialog window.

a. Choose one of the following two options :

(1) Select EXECUTE EXCEL to go directly to the Excel steel beam spreadsheet.

(2) Select SEND DATA TO FILE and enter a FILE NAME to defer design to a later date.

» **Note: This is the proper choice if you had difficulty in running Excel from CASM. Follow the procedure described in step L-2 of scheme 1A to pass the data on to Excel.**

b. Click on OK to continue.

2. Design the interior girder within the Excel design spreadsheet.

a. The Preliminary Design Spreadsheet will appear.

b. Use the scroll bar to view the nonvisible portions of the spreadsheet.

c. Use CARDFILE to review factors that may influence your decisions.

d. Using the Member pull-down menu, set the depth limit to 22.5 inches. The W16x40 is the lightest steel beam size displayed in the selection table.

» **Note: This selection has a self weight greater than estimated by 4 plf.**

(1) Select SELECT MEMBER from the Member pull-down menu to choose the desired beam size.

(2) Click on the selected beam in the list.

(3) Turn on SEND MEMBER SIZE TO CASM if the data was sent directly from CASM.

(4) Select OK to insert the beam designation in the CASM Beam Selection line in the lower part of the spreadsheet.

e. Select PRINT SPREADSHEET from the File pull-down menu to obtain a hardcopy of the spreadsheet.

f. Select RETURN TO CASM from the File pull-down menu. The Excel program will be closed.

g. If you activated Excel through CASM, reactivate the CASM program by selecting MAXIMIZE from the System menu if you are using a dual-screen workstation. Select RESTORE *if you are using the single-screen* workstation.

» **Note: If data were sent to a file, re-execute CASM and load your previous project data file.**

3. Proceed to the design of another element or revise the preliminary analysis of the current element as desired.

4. Select CANCEL in the Linear Elements dialog window to end work with the selected widely spaced element and to be able to proceed to the design of other element types.

G. Investigate the influence of a partition load perpendicular to the narrowly spaced elements in the center of the center bay.

1. Prepare a partition load based on the following components:

\>> Type over the previous exterior wall data and select ASSIGN.

Name : Partition

	Type	psf
Finish	: Gypboard 5/8"	3.1
Sheathing	:	0.0
Structure	: Stl Stud 16ga 4"@16	1.1
Insulation	:	0.0
Finish	: Gypboard 5/8"	3.1
Total	:	7.3

2. ASSIGN the load to the center of the center bay and perpendicular to the joist span. Set the wall height to 12 feet in the Wall Height dialog window.
3. Select OK to end working with wall dead loads.
4. Establish the parameters upon which analysis can be performed.
   a. Select material: STEEL.
   b. Select component from the Surface/Linear pull-down menu: NARROWLY SPACED : OPEN WEB JOISTS - K and click on one of the handles in the center bay.

   c. Review data in dialog windows and Guidelines.
   d. Continue on to Preliminary Analysis.
5. Preliminary Analysis of the open-web steel joist.

   a. Select load combination: Dead + Live.
   b. Select PRELIMINARY from the Design pull-down menu.
   c. Select units: FEET and POUNDS.
   d. Verify Load Combination of D+ L .
   e. Do not check APPLY LIVE LOAD REDUCTION.
   f. Do not check PATTERN OCCUPANCY LIVE LOAD.
   g. Do not check USE ACTUAL PROPERTIES.
   h. Turn off DL = DECK + SELF WEIGHT.
   i. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
   j. Select connectivity: simple span : HINGE and ROLLER.
   k. Estimate self weight: 0.0 plf.

\>> Note: Joist weight was smeared into the floor dead load.

   l. Review loads and connectivity displayed.

\>> The partition load is displayed as a concentrated load at midspan.

   m. Enter an appropriate analysis file name.

n. Review the shear, moment, relative deflection, loads and reaction diagrams on the display for any of the load cases shown in the window.

6. Preliminary Design of the open-web steel joist.

   a. Make appropriate selections within the Excel Data dialog window.

   b. Design the open-web steel joist within the Excel Preliminary Design spreadsheet.

      (1) Set depth limit: 24 inches.

      (2) Set deflection limits: Total L/240, Live L/360.

   c. Make member selection: 16K2

» **Note: The joist size is the same as without the concentrated partition load in scheme 1a and is at the deflection and allowable moment limit for this joist size.**

   d. Select RETURN TO CASM from the File pull-down menu.

7. Return to the CASM program.

H. Save the scheme 1b as filename: SCHEME1B.BLD.

[illegible]

[illegible]

A. OPEN the scheme 1a datafile name: SCHEME1A.BLD. The steps contained in A through D for scheme 1a are the same.

B. Draw Structure.

1. Select the DRAW STRUCTURE tool palette.

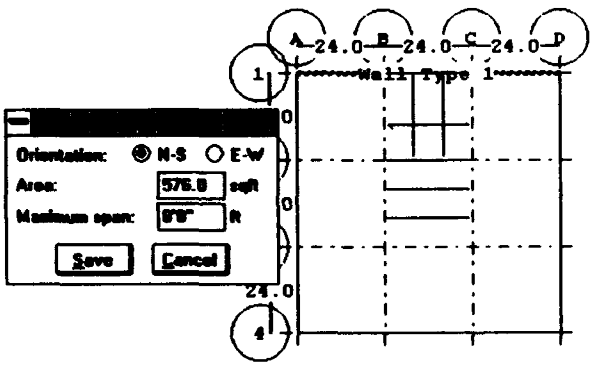
2. Delete all the narrowly spaced elements.

3. Insert widely spaced elements at third points spanning in the east/west direction within the center bay.

» Make sure DRAW SURFACE is not turned on if there already is a surface drawn in the bay.

4. Insert widely spaced elements at third points spanning in the north/south direction in the bay bounded by grid lines B-C and 1-2.

5. Insert a one-way surface above the beams.

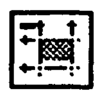


» Note: A surface must be drawn over widely spaced elements which are contained between grid lines since they are not assumed to have a surface transferring load to them.

C. Develop the independent load cases for this construction type.

1. Select the LOADS AND DESIGN tool palette.
2. Delete the Floor Type 1 Assigned dead load.
   a. Turn on SHOW DEAD loads to view current assigned dead loads.

» Note: Only displayed loads can be deleted.



   b. Select DELETE LOAD from the Edit pull-down menu. Handles appear at the centroids of all assigned dead loads.
   c. Select the appropriate handle to delete Floor Type 1 dead load. The load type will disappear from the display and handles will appear on any remaining dead loads to delete.

» Note: The Floor Type 1 dead load has only been removed from the building model. It still exists in the floor dead load list.

d. Double click the right mouse key to stop deleting loads.

3. Prepare the dead load based on the given load list and floor construction cross section.

Name	:Floor Type 2
Partition	:None
Finish	:Carpet & Pad
Deck	:MTL DK 3.0 / LTWT 2.5
Structure	:Steel Beams 0.0 psf
Mechanical	:3.0 psf
Electrical	:1.0 psf
Fire Protection	:Sprinklers Wet 4.0 psf
Ceiling	:Suspended Channel/Tile

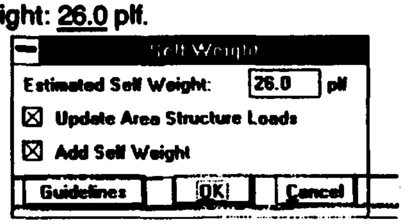
4. ASSIGN Floor Type 2 over the entire floor area.
5. Select STOP to end working with floor dead loads.
6. The given live load has not changed from scheme 1 and is still assigned to the entire floor.

D. **Establish element parameters necessary to design the typical third point beams within the center bay.**

1. Select material: STEEL.
2. Select component from the Surface/Linear pull-down menu: WIDELY SPACED : ROLLED SECTIONS and click on one of the handles in the center bay.
3. Review data in dialog windows and Guidelines.
4. Continue on to Preliminary Analysis.

E. **Preliminary Analysis of the third point beam.**

1. Select load combination: Dead + Live.
2. Select PRELIMINARY from the Design pull-down menu.
   a. Select units: FEET and KIPS.
   b. Verify Load Combination of D+ L .
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Turn off DL = DECK + SELF WEIGHT.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
3. Select connectivity: simple span : HINGE and ROLLER.
4. Estimate self weight: 26.0 plf.

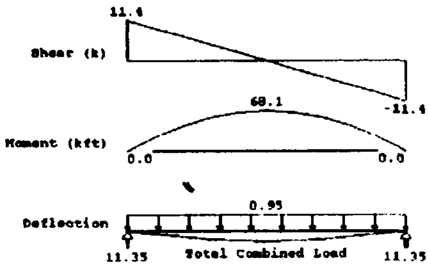


» **Note: Beam weight was not smeared into the floor dead load.**

5. Turn on UPDATE AREA STRUCTURE LOADS.

» **Note: This will smear the estimated beam weight into the floor type 2 area load.**

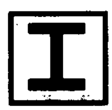
6. Review loads and connectivity displayed.
7. Enter an appropriate analysis file name.
8. Review the shear, moment, relative deflection, load and reaction diagrams on the display for any of the load cases shown in the window.



F. Preliminary Design of the third point beam.

1. Make appropriate selections within the Excel Data dialog window.
2. Design the third point beam within the Excel Preliminary Design spreadsheet.
   a. Set depth limit: 22.5 inches.
   b. Set deflection limits: Total L/240, Live L/360.
   c. Make member selection: W16x26.
   d. Select RETURN TO CASM from the File pull-down menu.
3. Return to the CASM program.

G. Establish element parameters necessary to design the typical interior column line girder along grid 2 between grids B and C.



1. Select material: STEEL.
2. Select component from the Surface/Linear pull-down menu: WIDELY SPACED : ROLLED SECTIONS and click on the handle on grid line 2.
3. Review data in dialog windows and Guidelines.
4. Continue on to Preliminary Analysis.

H. Preliminary Analysis of the typical interior girder.

1. Select PRELIMINARY from the Design pull-down menu.
   a. Select units: FEET and KIPS.
   b. Verify Load Combination of D+ L .
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Turn off DL = DECK + SELF WEIGHT.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.

2. Select connectivity: simple span : HINGE and ROLLER.

» **Note: The calculation of reactions of the perpendicular beams which frame into the girder are performed automatically. The self weight of the perpendicular beams is included in the updated floor type 2 area dead load.**

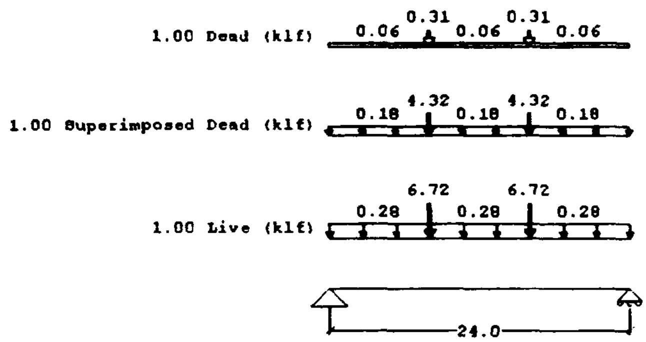
» **Note: Simple span connectivity assumptions are used when automatic calculation of beam reactions are performed.**

» **Note: The display will highlight the tributary areas concurrent with the calculation of beam reactions.**

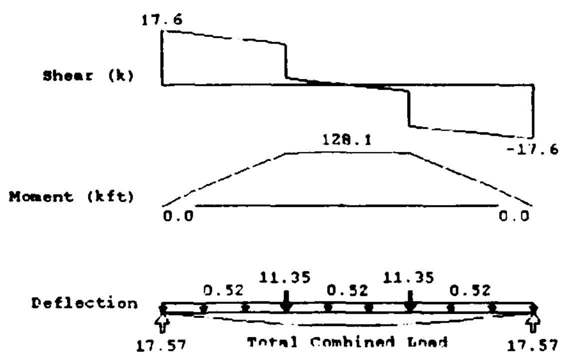
3. Estimate self weight: 45.0 plf.

» **Note: The estimated beam weights shown in the help window are based on uniform loads. No help is available for concentrated loads along a span, and an educated guess is required.**

4. Review loads and connectivity displayed.



5. Enter an appropriate analysis file name.
6. Review the shear, moment, relative deflection, load and reaction diagrams on the display for any of the load cases shown in the window.



I. **Preliminary Design of the typical interior girder.**
1. Make appropriate selections within the Excel Data dialog window.
2. Design the typical interior girder within the Excel Preliminary Design spreadsheet.
   a. Set depth limit: 22.5 inches.

b. Set deflection limits: Total L/240, Live L/360.

c. Make member selection: W18x40.

d. Select RETURN TO CASM from the File pull-down menu.

3. Return to the CASM program.

J. Save the scheme 2a as filename: SCHEME2A.BLD.

Solution - Scheme 2b Noncomposite Construction check method with continuous girder

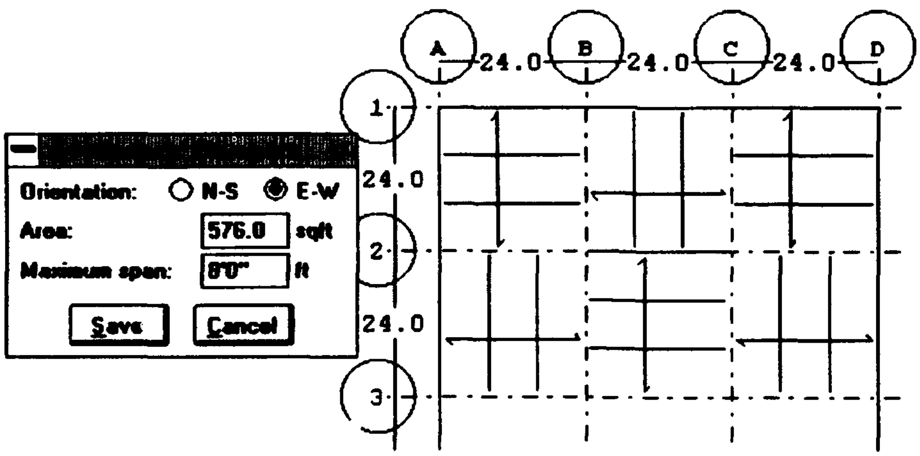
A. OPEN the scheme 2a datafile name: SCHEME2A.BLD. The steps contained in A through D for scheme 2a are the same.

B. Draw Structure.

1. Select the DRAW STRUCTURE tool palette.

2. Insert widely spaced elements at third points spanning in a checkerboard fashion to the left and to the right of the currently framed bays.

3. Insert a one-way surface above the beams within the four bays added in step 2.



4. Insert girders along grid line 2 between grids A to B and C to D.

» Note: You must insert girders between column lines (supports). Do not assign girder continuous from grid line A to D.

C. The same assigned independent load cases apply as for scheme 2A.

D. Establish element parameters necessary to design grid line 2 as a continuous girder for three spans.

1. Select material: STEEL.

2. Select component from the Surface/Linear pull-down menu: WIDELY SPACED : ROLLED SECTIONS and click on the left bay along grid line 2 between grids A and B.

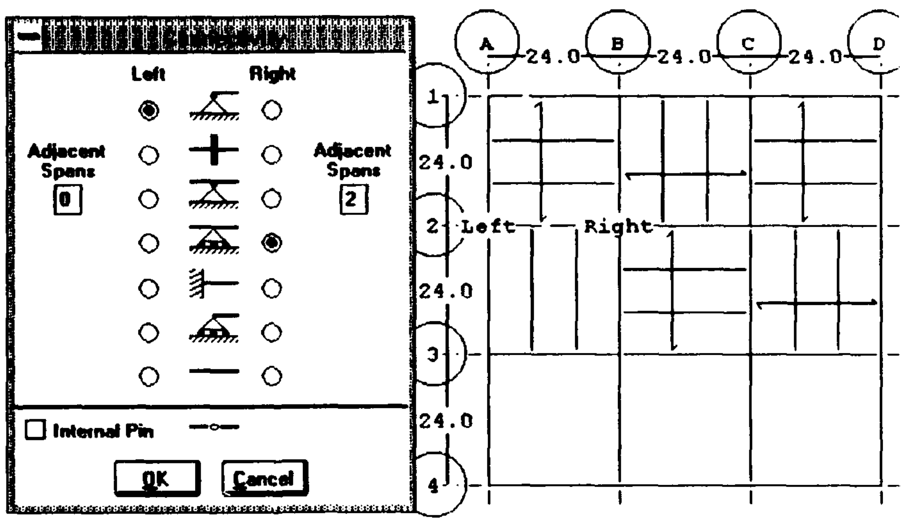
» **Note: The shears, moments and deflections of the initial span selected will be passed on to the preliminary design spreadsheet.**

» **Note: The left end span has been selected for analysis since it will produce the maximum negative and positive moments as well as the maximum deflection for a three-span continuous beam.**

3. Review data in dialog windows and Guidelines.
4. Continue on to Preliminary Analysis.

E. **Preliminary Analysis of the three-span continuous girder line.**

1. Select load combination: Dead + Live.
2. Select PRELIMINARY from the Design pull-down menu.
   a. Select units: FEET and KIPS.
   b. Verify Load Combination of D+ L .
   c. Do not check APPLY LIVE LOAD REDUCTION.
   d. Do not check PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Turn off DL = DECK + SELF WEIGHT.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
3. Select connectivity for the continuous three spans as follows:



   a. Select a hinge for the left end of the highlighted left span and a roller for a continuous beam at the right end.
   b. Set ADJACENT SPANS = 2 to the right side of the initial span selected.

» **Note: If the number of drawn adjacent spans is less than the number set, an error message will appear.**

» **Note: If the number of adjacent spans is set to zero, and you have selected a continuous support, CASM will search for a cantilevered support location.**

» **Note: The maximum total number of continuous spans that can be analyzed is four.**

c. Click on OK. A Connectivity dialog window will appear and a yellow dot will highlight the support location.

d. Select a CONTINUOUS BEAM ROLLER and click on OK. Another Connectivity dialog window will appear and another yellow dot will highlight the support location.

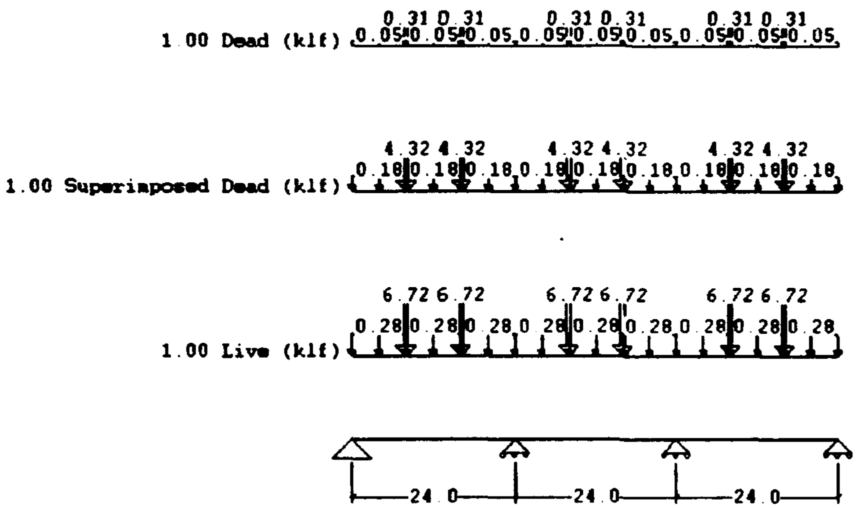
e. Select a ROLLER and click on OK.

4. Estimate girder self weight: 35.0 plf.

» **Note: The self weight help window applies to simple span members only. An educated guess is required for continuity situations.**

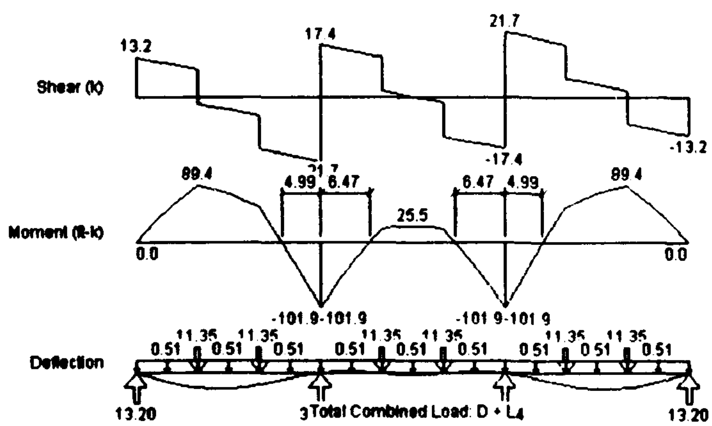
» **Note: The girder self weight was not smeared into the floor dead load.**

5. Review loads and connectivity displayed.



6. Enter an appropriate analysis file name.

7. Review the shear, moment and relative deflection diagrams on the display for any of the load cases shown in the window.



F. Preliminary Design of the exterior span of the continuous girder.

1. Make appropriate selections within the Excel Data dialog window.
2. Design the exterior bay girder within the Excel Preliminary Design spreadsheet.
   a. Set depth limit: 22.5 inches.
   b. Set deflection limits: Total L/240, Live L/360.
   c. Make member selection: W16X36.
   d. Select RETURN TO CASM from the File pull-down menu.
3. Return to the CASM program.

G. Save the scheme 2b as filename: SCHEME2B.BLD.

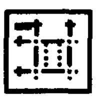
Solution Scheme 3: Cast In Place Concrete One Way Beam/Slab System

A. OPEN the scheme 2b datafile name: SCHEME2B.BLD. The steps contained in A through D for scheme 2b are the same.

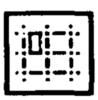
   a. Select SAVE AS from the FILE pull-down menu and rename the file as scheme3.bld.

B. Draw Structure.

1. Select the DRAW STRUCTURE tool palette.



2. Delete the third point beams that span in the north/south direction. This involves three bays.
3. Delete the surface element within those same three bays.
4. Insert widely spaced elements at third points spanning in the east/west direction within the three empty bays.



   a. Select the COPY STRUCTURE command from the Edit pull-down menu. Yellow handles will appear on structural members.
   b. Select the third point beams in bay A1- B2 with the left mouse key.
   c. Select the one-way surface in bay A1-B2 with the left mouse key.
   d. Double click the right mouse key to end selecting structural members to copy.
   e. Select grid location A1 as the base point to copy from with the left mouse key.

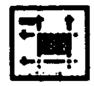
      The PASTE STRUCTURE command is automatically started.

   f. Select grid locations B1, A2, and C2 with the left mouse key.
   g. Double click the right mouse key to stop pasting structural members.

C. Develop the independent load cases for this construction type.

1. Select the LOADS AND DESIGN tool palette.
2. Delete the Floor Type 2 Assigned dead load.

a. Turn on SHOW DEAD loads to view current assigned dead loads.
b. Select DELETE LOAD from the Edit pull-down menu.
c. Select the appropriate handle to delete Floor Type 2 dead load.
d. Double click the right mouse key to stop deleting loads.

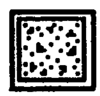


3. *Prepare the dead load based on the given load list and floor construction cross section.*

Name	:Floor Type 3
Partition	:None
Finish	:Carpet & Pad
Deck	:Concrete NLWT 4"
Structure	:Concrete Beams 0.0 psf
Mechanical	:3.0 psf
Electrical	:1.0 psf
Fire Protection	:None
Ceiling	:Suspended Channel/Tile

4. ASSIGN Floor Type 3 over the entire floor area.
5. Select OK to end working with floor dead loads.
6. The given live load has not changed from scheme 1 and is still assigned to the entire floor.

D. Establish element parameters necessary to design grid line 2 as a continuous girder for three spans.

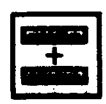


1. Select material: CONCRETE.
2. Select component from the Surface/Linear pull-down menu: WIDELY SPACED : BEAM-C.I.P. and click on the left bay along grid line 2 between grids A and B.

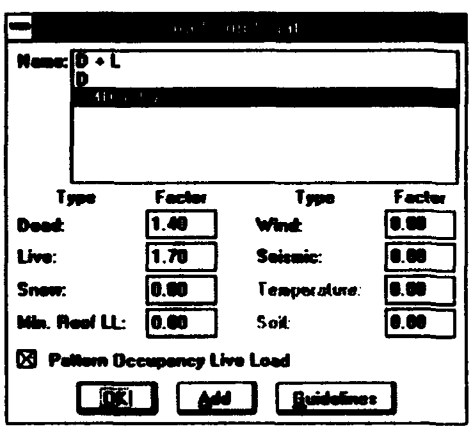
» Note: The left end span has been selected for analysis since it will produce the maximum negative and positive moments as well as the maximum deflection for a three-span continuous beam.

3. Review data in dialog windows and Guidelines.
4. Continue on to Preliminary Analysis.

E. Preliminary Analysis of the three-span continuous girder line.



1. Select load combination: 1.4 Dead + 1.7 Live.



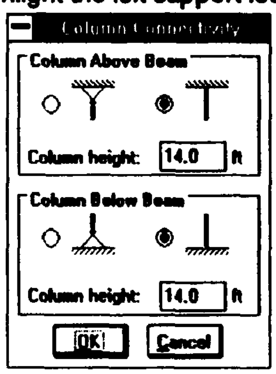
2. Turn on PATTERN OCCUPANCY LIVE LOAD in the Load Combinations dialog window and click on OK.

3. Incorporate live load reduction option.
   a. Select OCCUPANCY (LL) from the Loads pull-down menu.
   b. Select LLR GUIDELINES to review the code requirements regarding live load reductions.

» **Note: The tributary area for each beam span is less than 400 square feet and thus no reduction will actually be applied.**

   c. Click on OK to remove the Guidelines window.
   d. Turn on APPLY LIVE LOAD REDUCTION.
   e. Select STOP to end work with the live load list.

4. Select PRELIMINARY from the Design pull-down menu.
   a. Select units: FEET and KIPS.
   b. Verify Load Combination of 1.4D+ 1.7L .
   c. Select APPLY LIVE LOAD REDUCTION.
   d. Select PATTERN OCCUPANCY LIVE LOAD.
   e. Do not check USE ACTUAL PROPERTIES.
   f. Turn off DL = DECK + SELF WEIGHT.
   g. Do not check RE-ANALYZE ALL ADJOINING MEMBERS.
5. Select connectivity for the continuous three spans as follows:
   a. Select a FRAMED CONTINUITY support for the left and right end of the highlighted left span.
   b. Set ADJACENT SPANS = 2 to the right side of the initial span selected. Leave the left adjacent spans equal to zero.
   c. Click on OK. A Column Connectivity dialog window will appear and a yellow dot will highlight the left support location.



   d. Insert COLUMN HEIGHT = 14.0 feet for the Column Below Beam.

» **Note: This is necessary since the model does not contain a floor level below the one used for framing.**

   e. Set the end of the column above and below as FIXED.

f. Click on OK. A Column Connectivity dialog window will appear and a yellow dot will highlight the right support location.

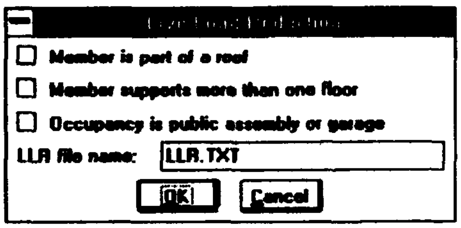
g. Repeat steps d through f for the right support of the span to be designed. The Connectivity dialog window will appear and a yellow dot will highlight the support location at the right end of the middle span.

h. Select a FRAMED CONTINUITY support and click on OK. The Column Connectivity dialog window will appear.

i. Repeat steps d through f for the support. The Connectivity dialog window will appear.

j. Select a FRAMED CONTINUITY support and click on OK. The Column Connectivity dialog window will appear.

k. Repeat steps d and e, then click on OK. A Live Load Reduction dialog window will appear.



6. Answer the Live Load Reduction questions which appear in the dialog window.

   a. Member is not part of a roof; therefore, do not select with an X.

   b. Member does not support more than one floor; therefore, do not select with an X.

   c. Occupancy is not a public assembly or garage; therefore, do not select with an X.

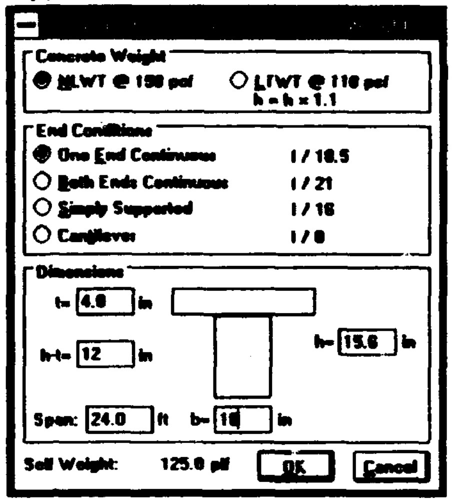
   d. Enter the desired LLR Filename as : LLR.TXT.

   e. Click on OK and load calculations begin. A Self Weight dialog window will appear.

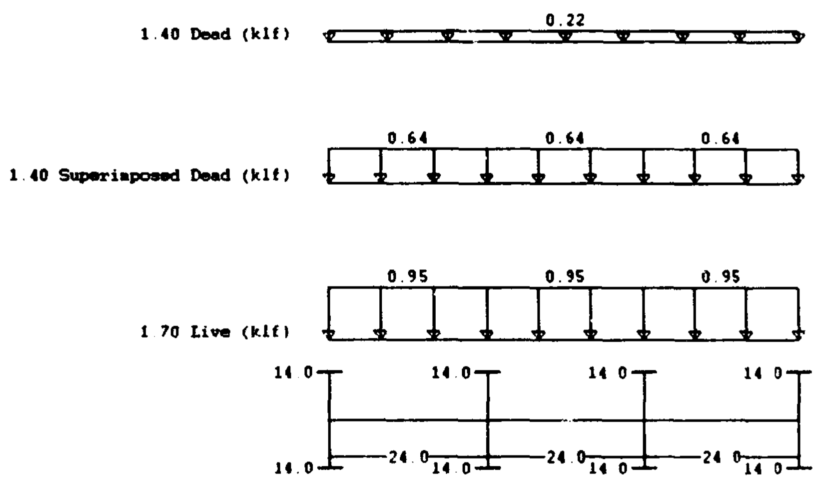
» **Note: The live load reduction calculation output file can be viewed and printed similar to that for wind and snow output. A sample of the output is listed below.**

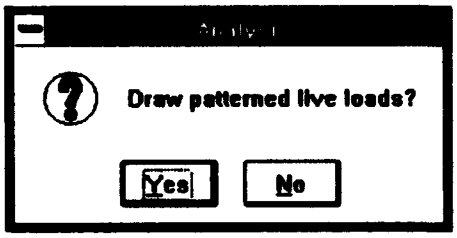
```
Project     : Corps Office Building
Location    : Vicksburg
Design Load : TM 5-809-1 1986
Time        : Thu Sep 19, 1991  2:03 PM

******************** Live Load Reduction *******************

Typical Floor
Offices w/smear corridor      (Lo) :   70.0 psf
Tributary area                (TA) :  192.0 sf
Area of influence (Ai) = 2*TA for beams.
Ai = 384.0 sf
Ai < 400.0 sf
No live load reduction taken.
L = Lo
+-----------------------+
|     L = 70.00 psf     |
+-----------------------+
```

7. Estimate girder self weight.
   a. Select Guidelines and a Concrete Beam Estimated Self Weight dialog window will appear.

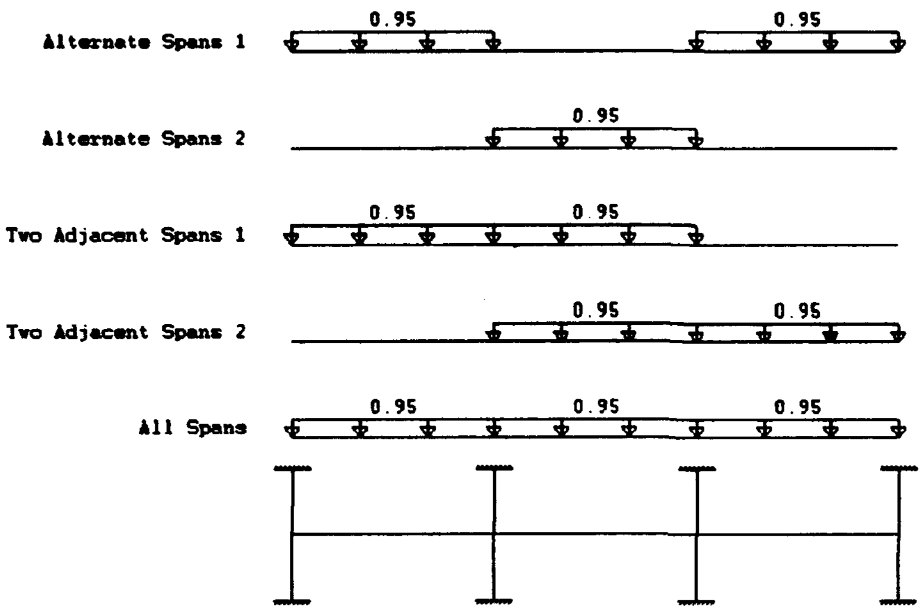


   b. Select NLWT @ 150 PSF.
   c. Select ONE END CONTINUOUS and the L/h ratio will become 18.5 in compliance with the minimum ratio required by the ACI Code to avoid deflection calculations.
   d. Enter the slab thickness t = 4.0 inches. The value of h-t will become 11.6 inches.
   e. Revise h-t= 12 inches.
   f. Revise the beam width b = 10.0 inches and pass the 125.0 plf self weight to the Self Weight dialog window.
   g. Click on OK to close the Concrete Beam dialog window.
   h. Click on OK to display the updated dead load plf diagram. A Draw Patterned Live Loads dialog window will appear.
8. Review the loads and connectivity displayed and select YES to draw the patterned live loads.



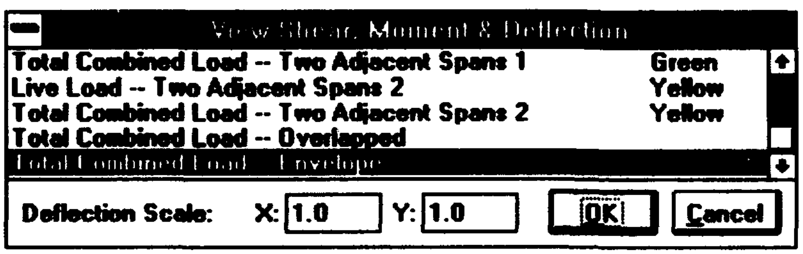


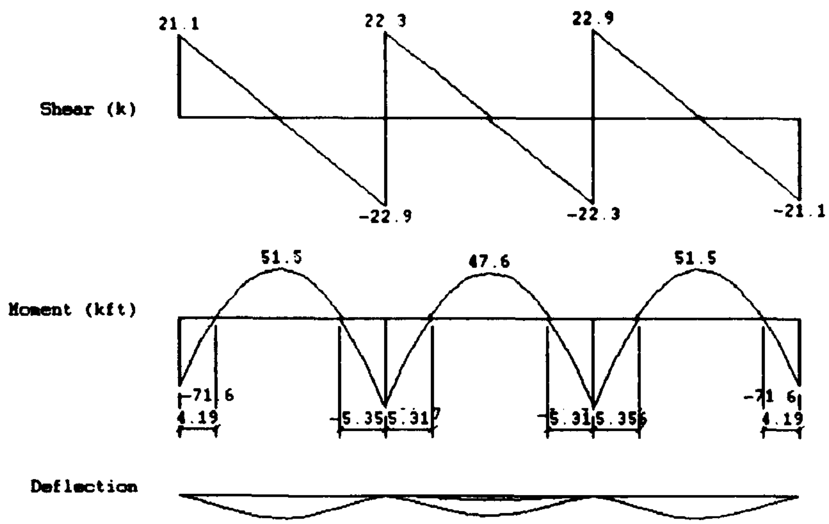
9. **Review the live load patterns displayed.**



10. **Enter an appropriate analysis file name and click on OK.**

11. **Review the shear, moment, relative deflection, loads and reactions diagrams on the display for any of the load cases shown in the window.**





Total Combined Load -- Envelope

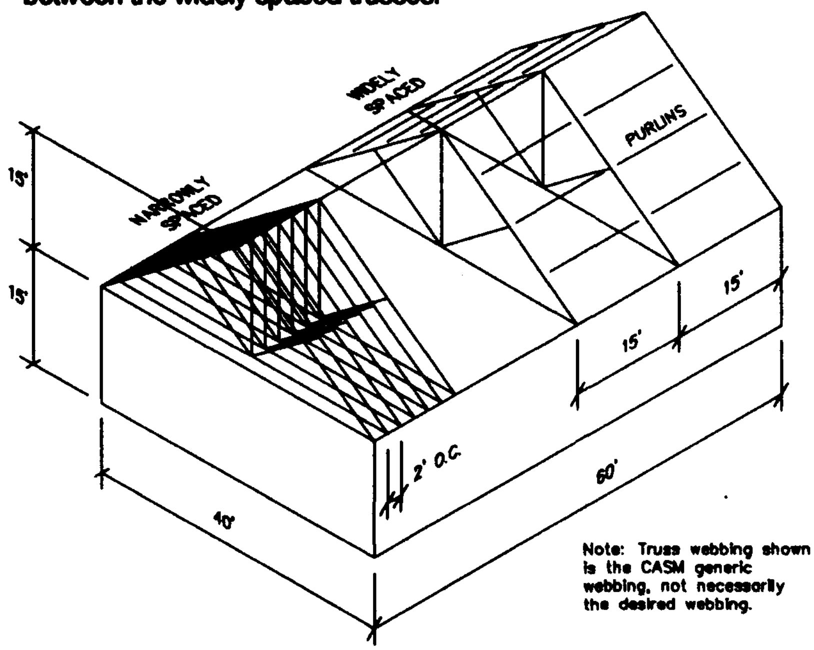
F. Preliminary Design of the exterior span of the continuous girder.

1. **Make appropriate selections within the Excel Data dialog window.**
2. **Design the exterior bay girder with in the Excel Preliminary Design spreadsheet.**
   a. **Set depth limit: 22.5 inches.**
   b. **Select Trial Beam Size: h= 16 inches; b= 10 inches. (Do not send to CASM).**
   c. **Select Rebar Size: left= #6; mid= #4; right= #6**
   d. **Send Trial Beam Size to CASM.**
   e. **Select RETURN TO CASM from the File pull-down menu.**
3. **Return to the CASM program.**

G. Save the scheme 3 as filename: SCHEME3.BLD.

■ Truss Design

Given: A one story gable roof building located at Westover AFB. Two framing options are to be reviewed. Therefore, there are 2'-0" o.c. narrowly spaced wood trusses across one half of the building and 15'-0" o.c. widely spaced steel trusses across the other half. Purlins are located at quarter points between the widely spaced trusses.



Dead Loads:

Narrowly spaced truss:

Top chord:	asphalt shingles 5/8" plywood
Bottom chord:	8" batt insulation 5/8" gypsum board

Widely spaced truss:

Top chord:	asphalt shingles 2" rigid insulation 1 1/2" - 20 ga. metal deck purlin weight
Bottom chord:	no superimposed dead loads

Required: Perform a preliminary analysis for a narrowly spaced truss element and a widely spaced truss element with purlins. Load cases are dead + snow and dead + wind for the narrowly spaced truss, and dead + snow for the widely spaced truss.

Solution:

A. Establish Criteria.

1. Select PROJECT and input the following data:

Project Name	: TRUSS EXAMPLE
City/Installation	: WESTOVER AFB
State	: MA
Design Load	: TM 5-809-1 1986

2. Select REGIONAL and check the following data:

Basic Wind Speed	: 75.0 mph
Coastal	: NO
Ground Snow Load	: 30.0 psf

3. Select SITE and input the following data:

Wind Importance	: I
Wind Exposure	: C
Snow Exposure	: C
Roof Slippery	: NO
Thermal Factor	: HEATED

B. Draw Volumetric Model

1. Select the DRAW MODEL tool palette.
2. Establish general layout requirements which are different than previously established.
   a. Use the following:

Define Units		: 12 inches
Snap To Units		: ON
Show Ground Plane		: ON
Ground Plane:		
Width	N-S	: 100 feet
	E-W	: 100 feet
Spacing	N-S	: 20 feet
	E-W	: 20 feet
Initial Shape Size:		
N-S		: 40 feet
E-W		: 60 feet
Height		: 15 feet
Orientation		: E-W
Stack On Last Shape		: ON
Directions Locked		: NONE

3. Place a CUBE on the ground plane with the required dimensions.

» **Note: If there is not a shape drawn, stack on last shape will draw the shape on the ground.**

4. Stack a PRISM on the cube to create the given gable roof height shown above.
5. Draw the roof structural elements.

a. Select an INCLINED STRUCTURAL PLANE from the View pull-down menu.

b. Select the DRAW STRUCTURE tool palette.

c. Select DEFINE GRID from the Grid/Opening pull-down menu.

N-S Spacing : 20 feet
E-W Spacing : 15 feet

d. Draw a widely spaced truss on grid line B.

(1) Select TRUSS-CUSTOM from the Surface/Linear pull-down menu.

(2) Define the area to draw the truss by selecting the handle on grid line B.

(3) Double click the right mouse key to end defining the area.

(4) Select SAVE in the Linear Elements dialog box to proceed to the next step in drawing the truss.

(5) Turn ON INCLUDE OPPOSITE SIDE OF ROOF in the Truss-Custom dialog box.

» Note: If Include Opposite Side of Roof is unchecked (off), then only half of a gable roof truss would be drawn.

(6) Select OK to store the truss.

e. Draw purlins supported by the widely spaced truss.

(1) Select WIDELY SPACED Linear from the Surface/Linear pull-down menu.

(2) Define the area to draw the purlins by selecting the handles on grid lines A and B.

(3) Double click the right mouse key to end defining the area. A single purlin will appear in the 2-D view.

(4) Set the following information in the Linear Elements dialog box:

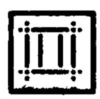
Orientation :E-W
Number of Elements :3 & Fixed (by placing an X in the box)

(5) Turn on DRAW SURFACE to draw this roof deck supported by the purlins.

(6) Select the RECALC button to redraw the new number of purlins.

(7) Select SAVE to store the purlins and deck.

f. Draw the ridge beam.



(1) Select WIDELY SPACED Linear from the Surface/Linear pull-down menu.

(2) Define the area to draw the ridge beam by selecting the handle between grid lines A and B.

(3) Double click the right mouse key to end defining the area.

(4) Select SAVE to store the ridge beam.

g. Draw the purlins, ridge beam and roof deck between grids B and C by using the Copy and Paste Structure commands.

(1) Select the Copy Structure command from the Edit pull-down menu. Yellow handles will appear on the structural elements.

(2) Select the ridge beam, purlins and deck handles with the left mouse key.

(3) Double click the right mouse key to end selecting structure to copy.

(4) Select grid location A2 with the left mouse key to define the base point to copy the structure.

The Paste Structure command is automatically started.

(5) Select grid location B2 with the left mouse key to select the base point from which the copied structure will be drawn.

(6) Double click the right mouse key to end pasting structure.

h. Draw the narrowly space trusses between grids C and E.

(1) Select TRUSS-CUSTOM from the Surface/Linear pull-down.

(2) Define the area to draw the trusses by selecting the following handles in order: between grid C3-D3, between grid C2-D2, and between grid E2-E3 if the south half of the roof is shown in 2-D. Otherwise, select handles between grids C1-D1, C2-D2, and E1-E2.

(3) Double click the right mouse key to end defining the area. There will be a particular number of evenly spaced trusses drawn dependent on the last number of drawn linear elements.

(4) Set the following information in the Linear Elements dialog box:

Orientation	: N-S
Spacing	: 2 feet & Fixed

(5) Turn on DRAW SURFACE.

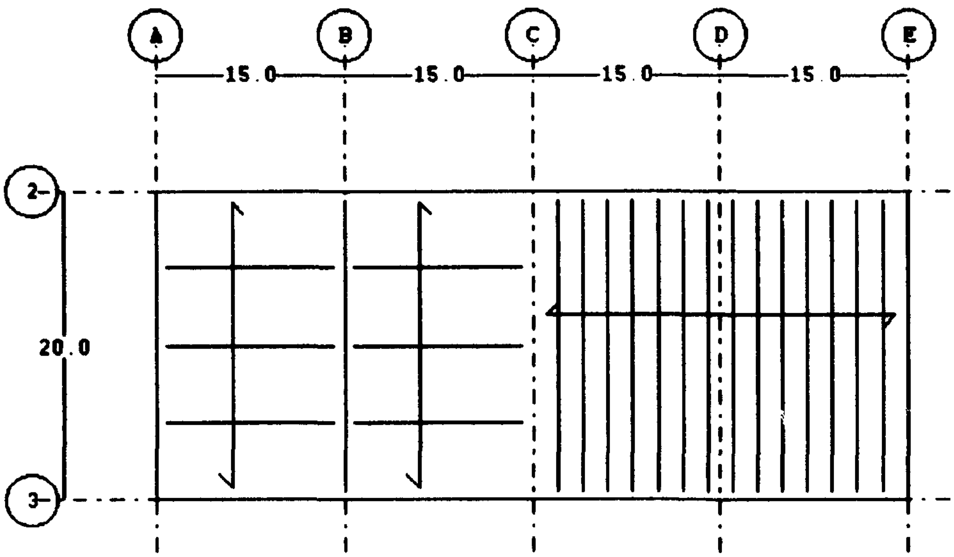
(6) Select the RECALC button to redraw the trusses at the new spacing.

(7) Select SAVE to proceed to the next step in drawing the trusses.

(8) Turn ON INCLUDE OPPOSITE SIDE OF ROOF in the Truss-Custom dialog box.

(9) Select OK to store the narrowly spaced truss.

» **Note: Structure needs to be drawn on or connecting to trusses so that loads are transferred to the truss correctly, i.e. uniform or concentrated. Trusses drawn less than 4'-0"o.c. do not assume a uniform load like narrowly spaced linear elements do.**

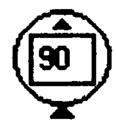


i. Draw the purlins, ridge beam and roof deck on the other half of the gable roof.

>> Note: The copy and paste structure commands cannot be used when desiring to copy structure onto a plane inclined in a different direction from the plane containing the structure to be duplicated.

(1) Select PERSPECTIVE (3D) from the View pull-down.

>> Note: Notice the trusses drawn in 3D. The webbing shown is the CASM generic webbing. The actual webbing desired is defined during the truss analysis procedure.



(2) Rotate the model 180 degrees.

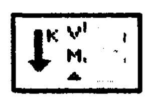
(3) Select the inclined structural plane opposite to the one previously selected.

(4) Draw the purlins, ridge beam and roof deck using similar steps above.

>> Note: The ridge beam needs to be draw on this half of the gable roof. When calculating loads, the beam is not aware of loads on the opposite roof plane. This quirk will be resolved in a future version of CASM.

>> Note: Save the model after completing many drawing steps. It is usually a good idea to periodically save the building model, especially before and after complicated procedures.

C. Develop the independent load cases.



1. Select the LOADS AND DESIGN tool palette.

2. Prepare the narrowly spaced truss TOP CHORD roof dead load from the given load list. Name the roof load type as NARROWLY TRUSS.

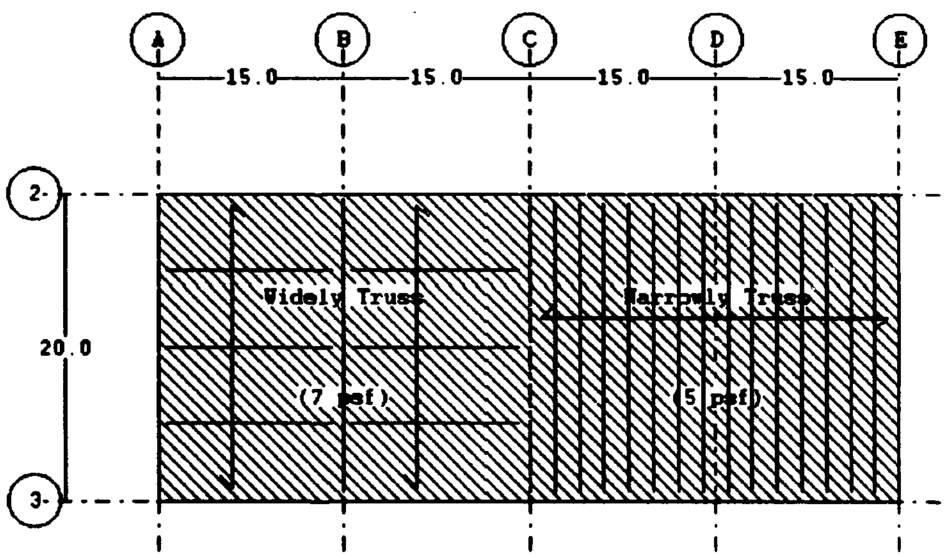
Roofing	:Asphalt Shingles
Deck	:5/8" Plywood
Structure	:(blank)
Mechanical	:None
Electrical	:None
Insulation	:None
Ceiling	:None

» Note: It is recommended to prepare only those loads you will assign to the current structural plane.

3. ASSIGN the wood construction roof dead load only above the narrowly spaced wood trusses.
4. Prepare the widely spaced truss top chord roof dead load from the given load list. Name the roof load type as WIDELY TRUSS.

Roofing	:Asphalt Shingles
Deck	:1 1/2" 20 ga Metal Deck
Structure	:Purlins 0.0 psf
Mechanical	:None
Electrical	:None
Insulation	:2" Rigid Insulation
Ceiling	:None

5. ASSIGN the steel construction roof dead load only above the widely spaced steel trusses.



6. Assign the roof dead loads to the opposite side of the gable roof.
   a. Select STOP to end working with roof dead loads.
   b. Select PERSPECTIVE (3D) from the Options pull-down menu.
   c. Rotate the model 180 degrees.


d. Select the INCLINED STRUCTURAL PLANE opposite to the one previously selected.
e. Select ROOF (DL) from the Loads pull-down menu.
f. ASSIGN the Widely Truss roof dead load.
g. Select the Narrowly Truss roof dead load from the name drop down list.
h. ASSIGN the Narrowly Truss roof dead load.
i. Select STOP to end working with roof dead loads.

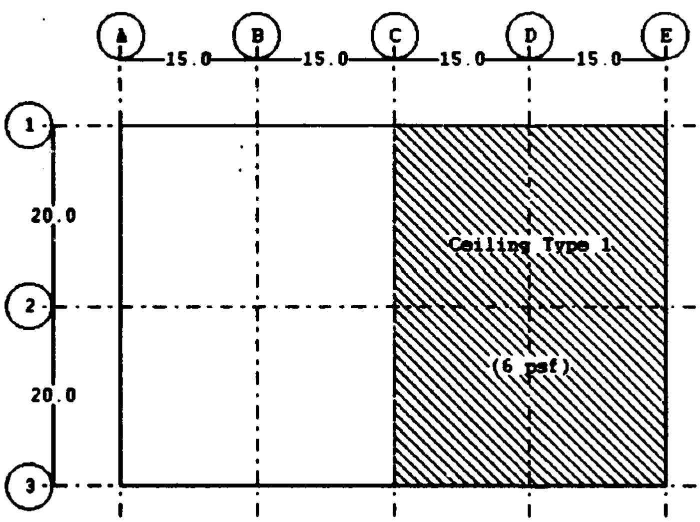
7. Prepare the narrowly spaced truss bottom chord dead load.



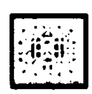
a. Select PERSPECTIVE (3D) from the View pull-down menu.
b. Select the ceiling HORIZONTAL STRUCTURAL PLANE.
c. Select CEILING (DL) from the Loads pull-down menu.
d. Fill in the appropriate bottom chord dead loads and name the ceiling load type as BOTTOM CHORD.

Mechanical	:None
Electrical	:None
Insulation	:8" Batt Insulation
Structure	:(blank)
Ceiling	:5/8" Gypsum Board

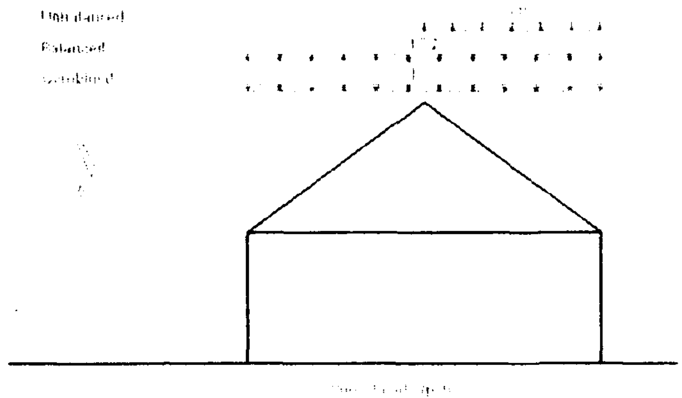
e. ASSIGN the ceiling dead load in the area beneath the narrowly spaced trusses, between grid locations C1, E1, E3, and C3.



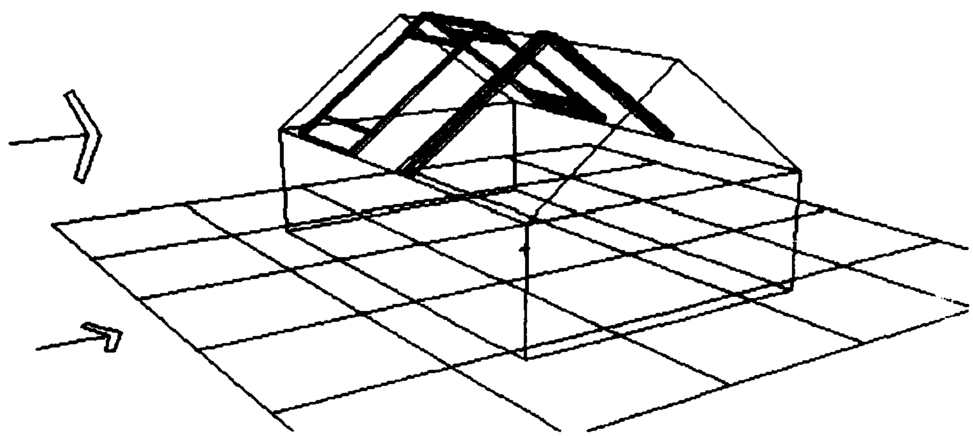
f. Select STOP to end working with ceiling dead loads.



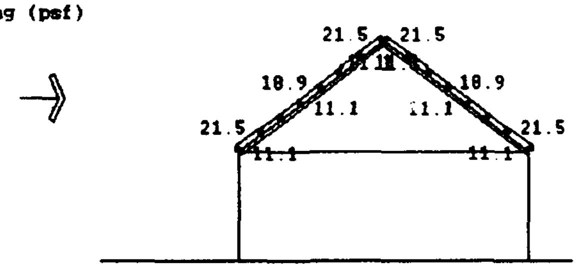
8. Calculate Snow Loads.
a. You should be an expert at this procedure by now.



9. **Calculate Components and Cladding Wind Loads.**
   a. **Define the truss tributary area on a typical narrowly spaced truss and a typical widely spaced truss. Name the components and cladding loads as: Narrowly Spaced Truss and Widely Spaced Truss.**
   b. **Prove your expertise by completing the procedure on your own. If you need help, follow Wind: Components and Cladding: Example One: step D.**



Wind Load: Components & Cladding (psf)



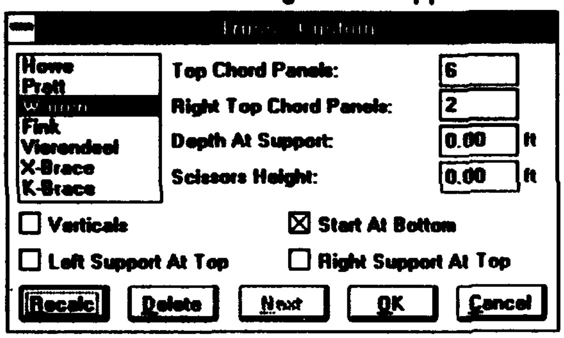
» **Note: Save model.**

D. **Establish element parameters necessary to analyze a typical narrowly spaced truss.**
   1. **Select material: <u>WOOD</u> from the Mat'l pull-down.**

2. Select either INCLINED STRUCTURAL PLANE.
3. Select component from the Surface/Linear pull-down menu: TRUSS-CUSTOM and click on the same narrowly spaced truss for which the wind components and cladding tributary area was defined.
4. Review the data in the dialog windows and Guidelines.
5. Continue on to Preliminary Analysis.

E. Preliminary Analysis of the narrowly spaced truss.

1. Select the load combination: DEAD + SNOW.
2. Select PRELIMINARY from the Design pull-down menu.
3. Select Units options.
   a. Select units of FEET and POUNDS.
   b. Verify load combination of D+ S.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
4. Select connectivity as a HINGE and a ROLLER. A 2D elevation view of the truss and the Truss-Custom dialog box will appear.



5. View the various truss webbing configurations.
   a. Set the TOP CHORD PANELS as 8.

>> Note: The right top chord panels are disabled when a gable truss is symmetrical.

   b. Select RECALC to redraw the truss with 8 panels.
   c. Select HOWE to draw the truss in the howe configuration.
   d. Select WARREN to draw the truss in the warren configuration.
   e. Turn ON VERTICALS to add vertical web members to the warren truss and select RECALC.
   f. Turn ON START AT BOTTOM to flip the diagonal members and select RECALC.
   g. Select FINK to draw the truss in the fink configuration.
   h. Select VIERENDEEL to draw the truss with no diagonals and all moment connections.
   i. Set the DEPTH AT SUPPORT as 5 feet and select RECALC.

j. Select X-BRACE to draw the truss with double diagonals.

k. Select K-BRACE to draw the webs in the K configuration.

l. Turn OFF START AT BOTTOM to flip the K direction.

m. Turn ON LEFT SUPPORT AT TOP and RIGHT SUPPORT AT TOP and select RECALC.

n. Turn OFF LEFT SUPPORT AT TOP and RIGHT SUPPORT AT TOP and select RECALC.

o. Set DEPTH AT SUPPORT to 0 feet, SCISSORS HEIGHT to 5 feet, and the TOP CHORD PANELS as 4 then select HOWE. The truss will be drawn as a scissors truss.

6. Select the required truss type and webbing configuration.

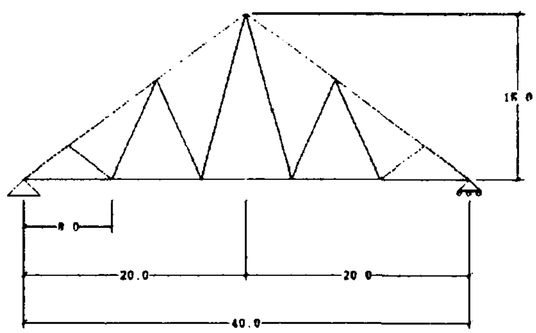
   a. Set the TOP CHORD PANELS as 6.

   b. Turn ON START AT BOTTOM.

   c. Turn OFF VERTICALS.

   d. Set the DEPTH AT SUPPORT and SCISSORS HEIGHT to 0 feet.

   e. Select the WARREN webbing configuration.



   f. Select OK and the loads will be applied to the truss.

7. Select the truss self weight.

   a. Select the GUIDELINES button for an estimated member self weight. The estimated weights of a wood truss spanning 40 feet will appear.

   b. Calculate the estimated self weight as 3.25 plf times 2 feet o.c. which equals 6.5 plf.

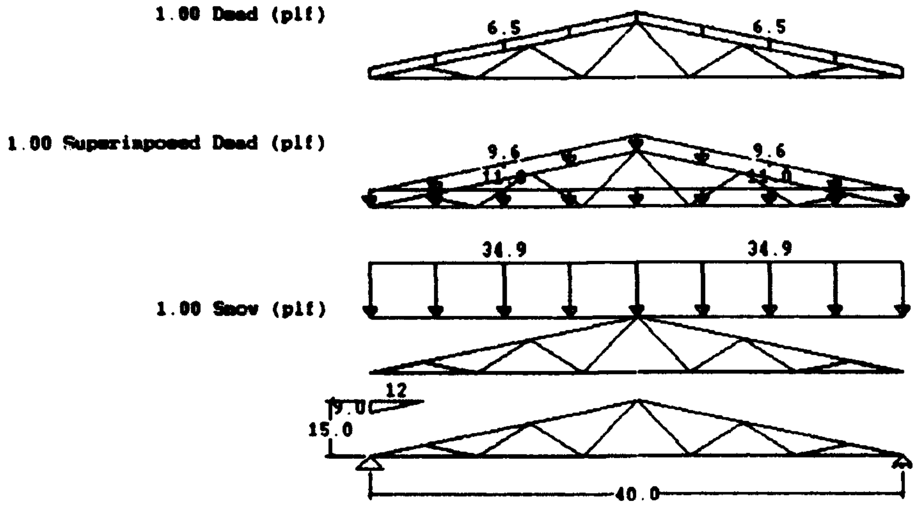
   c. Select CLOSE to close the estimated member weight dialog box.

   d. Set the ESTIMATED SELF WEIGHT as 6.5 plf.

   e. Select OK and the updated loads will appear.

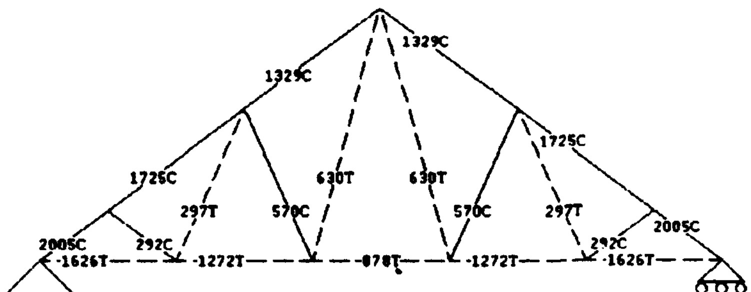
8. Review the loads and connectivity displayed. Enter an appropriate analysis file name and select YES. The analysis will be performed and its progress is shown.

9. Review the axial member forces, deflection, and loads and reactions diagrams on the display. View the node and element numbers used in the analysis. The member lengths are the lengths of each analysis member.

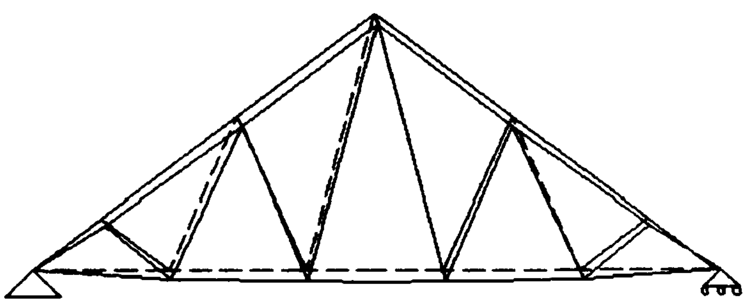


» **Note: Color is used to indicate relative magnitude of axial force in the following progression:**

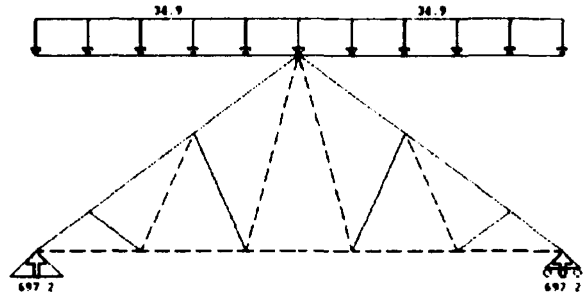
red = maximum
yellow
cyan
blue = minimum.



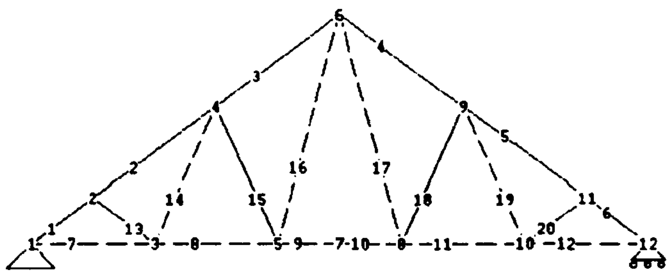
Total Combined Load -- Axial (lb)



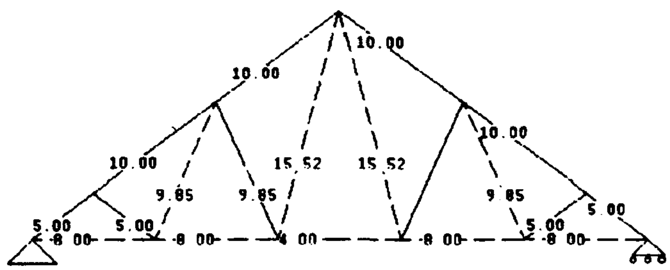
Total Combined Load — Deflection



Snow Load -- Loads & Reactions (lb)

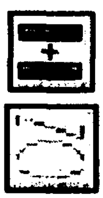


Node and Element Numbers



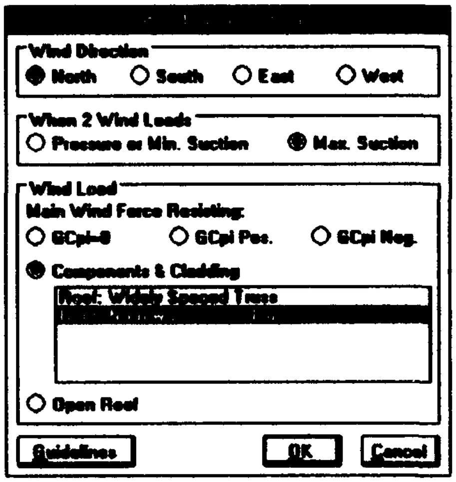
Member Lengths (ft)

10. Select CANCEL since there is no truss member design at this time.
11. Reanalyze the truss with the load combination dead + wind.



   a. Select load combination: DEAD + WIND.
   b. Select PRELIMINARY from the Design pull-down menu.
   c. Select Units options.
      (1) Select units of FEET and POUNDS.
      (2) Verify load combination of D+ S.
      (3) Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK + SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
   d. Select connectivity as a HINGE and a ROLLER.
   e. Select the required truss type and webbing configuration.

f. Set the following wind load options.



(1) Select the Wind Direction as NORTH.

(2) Turn ON MAX. SUCTION for When 2 Wind Loads.

» Note: When 2 Wind Loads refers to a pressure and a suction value appearing on the same surface. It is necessary to select one or the other for analysis.

(3) Select COMPONENTS & CLADDING for the Wind Load.

(4) Select ROOF: NARROWLY SPACED TRUSS for the name of the components and cladding load to use.

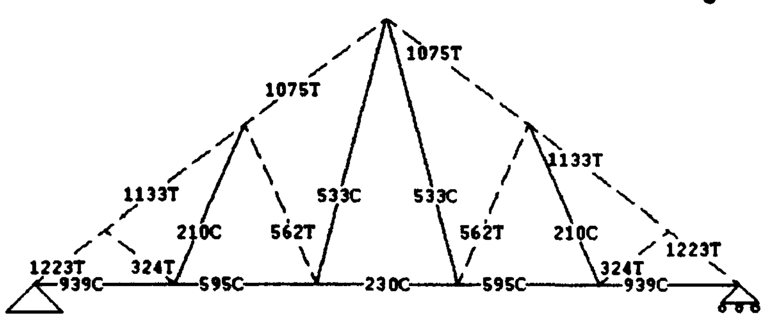
(5) Click on OK.

g. Select the truss self weight as 6.5 plf.

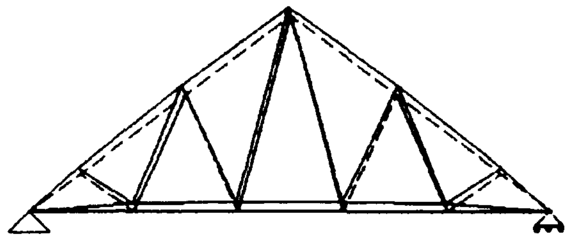
h. Enter an appropriate analysis file name.

i. Review the axial member forces, deflection, and loads and reactions diagrams.

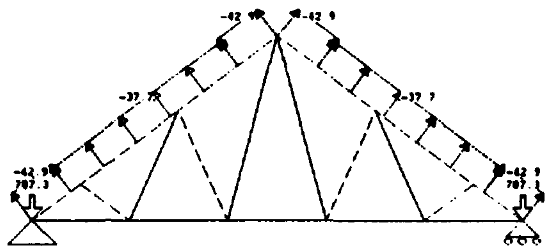
j. Select CANCEL since there is no wood truss member design at this time.



Wind Load -- Axial (lb)



Wind Load -- Deflection



Wind Load – Loads & Reactions

12. Select CANCEL in the Linear Elements dialog box.

F. Establish element parameters necessary to analyze the widely spaced truss.

1. Select material: STEEL.

2. Analyze a purlin to smear the purlin dead load weight into the uniformly distributed roof dead load.
   a. Select a widely spaced component from the Surface/Linear pull-down menu: 'C' CHANNELS and select a purlin.
   b. Select PRELIMINARY from the Design pull-down menu.

   c. Select Units options.
      (1) Select units of FEET and POUNDS.
      (2) Verify load combination of D+ S.
      (3) Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
   d. Select connectivity as a HINGE and a ROLLER.
   e. Set the ESTIMATED SELF WEIGHT as 10.0 plf and turn ON UPDATE AREA STRUCTURE LOADS and click on OK to proceed.
   f. Review the loads and connectivity displayed. Enter an appropriate analysis file name and select YES. The analysis will begin.
   g. Review the shear, moment, deflection, loads and reactions diagrams on the display.
   h. Select CANCEL since we are not going to design the member to check our estimate.

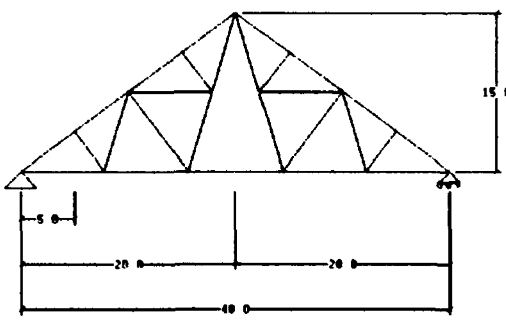
>> Note: There currently is no 'C' channel design spreadsheet available.

i. Select CANCEL in the Linear Elements dialog box to end analysis of the purlin.

3. Select truss component from the Surface/Linear pull-down menu: TRUSS-CUSTOM and click on the widely spaced truss located on grid line B.
4. Review the data in the dialog windows and Guidelines.
5. Continue on to Preliminary Analysis.

G. Preliminary Analysis of the widely spaced truss.

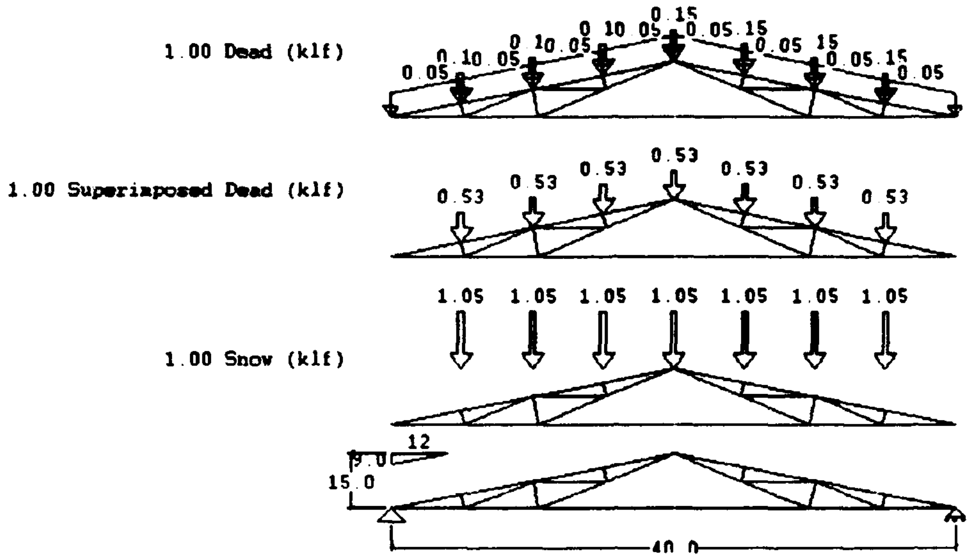
1. Select PRELIMINARY from the Design pull-down menu.
2. Select Units options.
   a. Select units of FEET and POUNDS.
   b. Verify load combination of D+ S.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
3. Select connectivity as a HINGE and a ROLLER. A 2D elevation view of the truss and the Truss-Custom dialog box will appear.
4. Select the truss webbing configuration.
   a. Set the TOP CHORD PANELS as 8.
   b. Turn OFF START AT BOTTOM.
   c. Turn OFF VERTICALS.
   d. Set the DEPTH AT SUPPORT and SCISSORS HEIGHT to 0 feet.
   e. Select the FINK webbing configuration.



   f. Select OK and analysis will begin. The analysis of each purlin which frames into the truss will be performed to generate the concentrated loads. When finished, the Self Weight dialog box will appear.
5. Select the truss self weight.
   a. Select the GUIDELINES button for an estimated member self weight. The estimated weights of a steel truss spanning 40 feet will appear.
   b. Calculate the estimated self weight as 3 plf times 15 feet o.c. which equals 45 plf.
   c. Select CLOSE to close the estimated member weight dialog box.
   d. Set the ESTIMATED SELF WEIGHT as 45 plf.
   e. Turn ON ADD SELF WEIGHT.

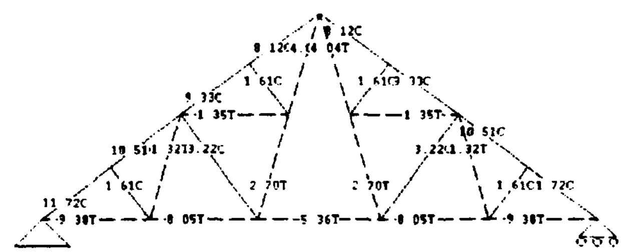
f. Select OK and the updated loads will appear.

6. Review the loads and connectivity displayed. Enter an appropriate analysis file name and select YES.

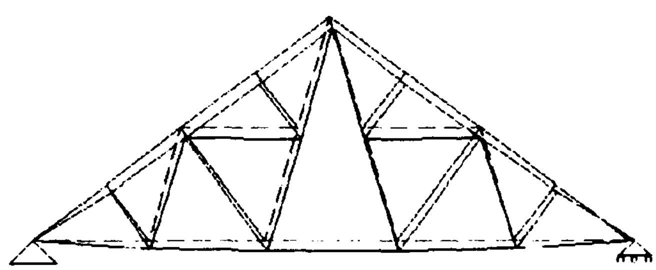


» **Note: When concentrated loads do not fall on panel points, the user has two options: 1. to continue with the analysis or 2. cancel the analysis by selecting NO from the Analysis dialog window. Re-select PRELIMINARY ANALYSIS to revise the truss configuration.**

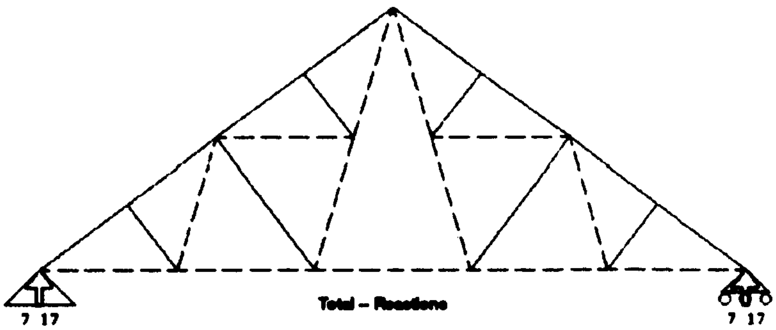
7. Review the axial forces, deflection, and loads and reactions diagrams on the display. View the node and element numbers used in the analysis. The member lengths are the lengths of each analysis member.



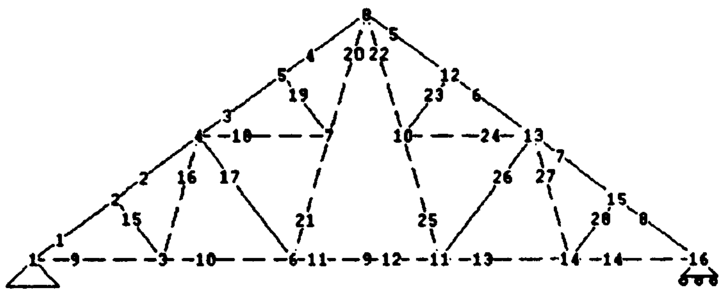
Total Combined Load -- Axial (k)



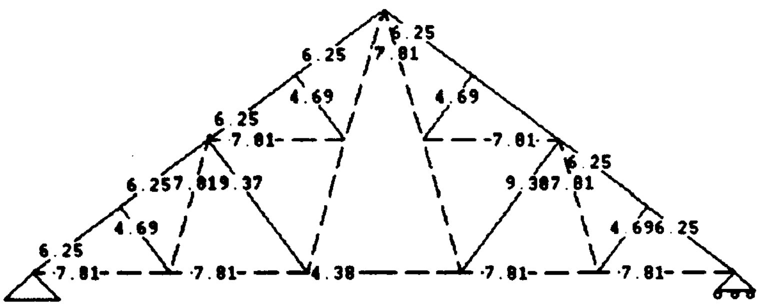
Total Combined Load -- Deflection



Total – Reactions



Node and Element Numbers



Member Lengths (ft)

8. Select <u>OK</u> to design truss members with Excel. The member design is limited to the top chord member, bottom chord member, maximum tension member, and maximum compression member. The sizes can be passed to CASM for a more accurate truss analysis.
9. Select <u>CANCEL</u> in the Linear Elements dialog box.

H. Save the model as: TRUSS1.BLD.

■ Column Design

Given: A three story plus basement steel framed administrative building on a 20 foot square grid, located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico. Floor to floor heights are assumed to be 14 feet.

Occupancy Live Loads:

First Floor:	Public circulation:	100 psf
Second & Third Floor:	Offices:	50 psf

Dead Loads:

Floor:
- Partition 101-200 plf
- Carpet & Pad
- Metal Deck 1.5" NLWT 2.5"
- Steel Beams - 4.0 psf non-composite
- Mechanical A/C Ducts - 3.0 psf
- Electrical/Lighting - 2.0 psf
- Suspended Channel - Tile Ceiling

Roof:
- Composite 5-ply/Gravel roofing
- Steel 1-1/2" 20 gauge deck
- Steel Bar Joist 24' at 4' o.c.
- Mechanical A/C Ducts - 3.0 psf
- Electrical/Lighting - 2.0 psf
- Rigid Roof Insulation 4"
- Suspended Channel - Tile Ceiling

Required: Perform a column load rundown for the center, edge and corner column. Use the dead + live + min. roof live load case. Calculate column loads without and with live load reductions.

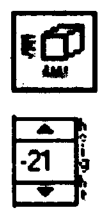
Solution:

A. Establish Criteria.

1. Re-open model from wind tutorial example two. If unavailable, complete step A of the example.

B. Draw Volumetric Model.

1. If 3D model is not drawn, complete step B of wind example two.
2. Draw a basement level.
   a. Set the INITIAL SHAPE SIZE HEIGHT as 14 feet.
   b. Turn ON STACK ON PLANE.
   c. Adjust the viewing height to a worms eye view of the underside of the ground floor plane.
   d. Select CUBE.



» **Note: If you opened wind example two, wind loads will have to re-calculated since you are altering the basic geometry. For this example, we will not need wind loads.**

e. Select the handle in the center of the ground floor plane.

f. Click the left mouse key to set the cube.

g. Double click the right mouse key to end stacking cubes on a plane.

h. Re-position the perspective to a bird's eye view.

3. Draw structural columns.

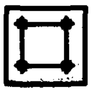
   a. Select the second floor HORIZONTAL STRUCTURAL PLANE. A 2-D view of the plane will appear.

   b. Select the DRAW STRUCTURE tool palette.

   c. Select DEFINE GRID from the Grid/Opening pull-down menu.

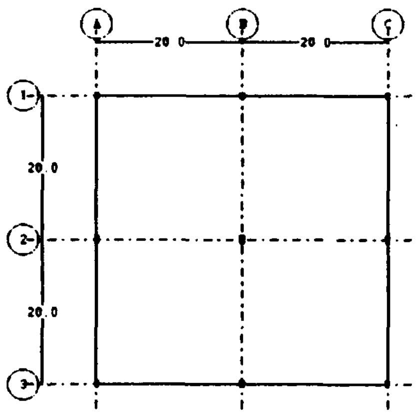
      N-S Spacing : 20 feet
      E-W Spacing : 20 feet

   d. Turn ON FOOTINGS from the Column/Wall pull-down menu. Footings will then be drawn at the bottom of each column when the columns are drawn.



   e. Draw columns on all grid intersections.

      (1) Select Column ALL GRID INTERSECTIONS from the Column/Wall pull-down menu. Columns and their respective footings will appear.



      (2) Select either a N-S or an E-W orientation.

      (3) Turn ON ALL FLOORS to duplicate all columns on all floor levels.

      (4) Select SAVE to store the columns on all floors.

» **Note: No other structure is necessary to be drawn since column and wall load rundowns are calculated based on a tributary area to the next column, wall or edge.**

» **Note: You may wish to save the model now.**

C. Develop Independent Load Cases.

   1. Select the LOADS AND DESIGN tool palette.

   2. Prepare the floor dead load from the given load list.

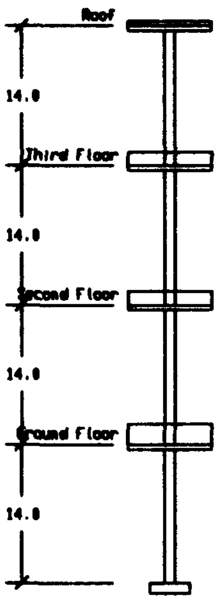
3. Turn ON ASSIGN ALL FLOORS.
4. ASSIGN the floor dead load on the entire floor plane.
5. Turn OFF ASSIGN ALL FLOORS.
6. Prepare the 50 psf Office occupancy live load list.
7. ASSIGN the live load on the entire floor plane.
8. Rename the second floor plane to SECOND FLOOR in the STRUCTURAL PLANE INFORMATION dialog box.
9. Switch to the third floor HORIZONTAL STRUCTURAL PLANE.
10. ASSIGN the 50 psf office live load on the entire floor plane.
11. Rename the third floor plane to THIRD FLOOR in the STRUCTURAL PLANE INFORMATION dialog box.
12. Switch to the ground floor HORIZONTAL STRUCTURAL PLANE.
13. Prepare the 100 psf Public and Circulation occupancy live load.
14. ASSIGN the live load on the entire floor plane.
15. Rename the ground floor plane to GROUND FLOOR in the STRUCTURAL PLANE INFORMATION dialog box.
16. Switch to the roof HORIZONTAL STRUCTURAL PLANE.
17. Prepare the roof dead load from the given load list.
18. ASSIGN the roof dead load on the entire roof plane.
19. Rename the roof plane to ROOF in the STRUCTURAL PLANE INFORMATION dialog box.

» Note: Save model.

D. Establish element parameters to perform the column load rundown for column B2.
   1. Select material: STEEL.
   2. Select component from the Column/Wall pull-down menu: Column, ROLLED SECTIONS and click on column B2.
   3. Review the data in the dialog windows and Guidelines.
   4. Continue on to the Preliminary Analysis.

E. Preliminary Analysis of column B2.
   1. Select the load combination: DEAD + LIVE + MIN. ROOF.
   2. Select PRELIMINARY from the Design pull-down menu.
   3. Select Units options:
      a. Select units of FEET and KIPS.
      b. Verify load combination of D + L + Lr.
      c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
   4. Enter a minimum Roof Live Load calculation output file.
   5. Select to use the Tributary Area Method of calculating the loads at each level.

6. Set the column self weight as 75 plf and select OK.
7. Review the column load rundown.



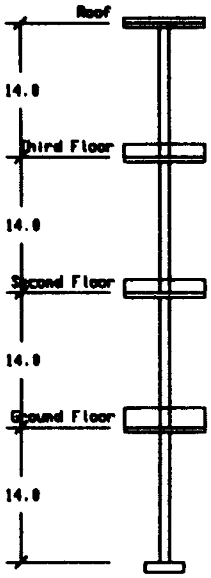
Tributary Area	Self Weight	DL	LL	Lr	TL	Sum DL	Sum LL	Sum Lr	Sum TL
400.0		8.4	0.0	6.4	14.8				
	1.1					9.4	0.0	6.4	15.8
400.0		24.0	20.0	0.0	44.0				
	1.1					34.5	20.0	6.4	60.9
400.0		24.0	20.0	0.0	44.0				
	1.1					59.5	40.0	6.4	105.9
400.0		24.0	40.0	0.0	64.0				
	1.1					84.6	80.0	6.4	171.0

Column B-2 Load Run Down (k)

8. The user has three options at this point.
   a. Design the member in Excel.
   b. Send the data to a file to design the column at another time.
   c. Select CANCEL to proceed without designing the column.
9. Make appropriate selection within the Excel Data dialog window.
10. Design column B-2 within the Excel Preliminary Design spreadsheet.
    a. Set depth limit to 16 inches.
    b. Make a member selection, W8x40 Levels 1 & 2; W8x24 Levels 3 & 4.
    c. Send Trial Column Size to CASM.
    d. Select RETURN TO CASM from the file pull-down menu.
11. Return to the CASM program.

F. Preliminary Analysis of column A1.

1. Perform steps D and E selecting column A1.

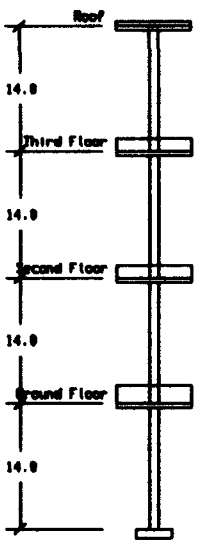


	Tributary Area	Self Weight	DL	LL	Lr	TL	Sum DL	Sum LL	Sum Lr	Sum TL
Roof	100.0		2.1	0.0	2.0	4.1				
		1.1					3.1	0.0	2.0	5.1
Third Floor	100.0		6.0	5.0	0.0	11.0				
		1.1					10.2	5.0	2.0	17.2
Second Floor	100.0		6.0	5.0	0.0	11.0				
		1.1					17.2	10.0	2.0	29.2
Ground Floor	100.0		6.0	10.0	0.0	16.0				
		1.0					24.3	20.0	2.0	46.3

Column A-1 Load Run Down (k)

G. Preliminary Analysis of column A2

1. Perform steps D and E selecting column A2.

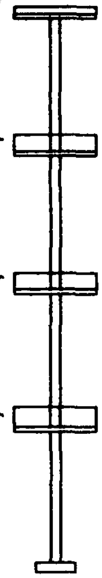


	Tributary Area	Self Weight	DL	LL	Lr	TL	Sum DL	Sum LL	Sum Lr	Sum TL
Roof	200.0		4.2	0.0	4.0	8.2				
		1.1					5.2	0.0	4.0	9.2
Third Floor	200.0		12.0	10.0	0.0	22.0				
		1.1					18.3	10.0	4.0	32.3
Second Floor	200.0		12.0	10.0	0.0	22.0				
		1.1					31.3	20.0	4.0	53.3
Ground Floor	200.0		12.0	20.0	0.0	32.0				
		1.1					44.4	40.0	4.0	88.4

Column A-2 Load Run Down (k)

H. Preliminary Analysis of column B2 with live load reduction.

1. Turn ON APPLY LIVE LOAD REDUCTION in the Occupancy Live Load dialog box.
2. Perform steps D and E selecting column B2 and enter an appropriate live load reduction file name when asked.



	Tributary Area	Self Weight	DL	LLR	LLR	LLR	LLR	Lr	TL	Sum DL	Sum LLR	Sum Lr	Sum TL
Roof	400.0		8.4	0.0	0.0	0.0	0.0	6.4	14.8				
14.0		1.1								9.4	0.0	6.4	15.8
Third Floor	400.0		24.0		12.5	10.3	9.3	0.0	36.5				
14.0		1.1								34.5	12.5	6.4	53.4
Second Floor	400.0		24.0			10.3	9.3	0.0	34.3				
14.0		1.1								59.5	20.6	6.4	86.6
Ground Floor	400.0		24.0				10.7	0.0	42.7				
14.0		1.1								84.6	37.3	6.4	128.3

Column B-2 Load Run Down (k)

3. Review output data and note that live load reduction influences the axial loads on columns B2.

l. Save the model as: COLUMN1.BLD.

■ Wall Design

[illegible]

Given: A three story plus basement steel framed administrative building on a 20 foot square grid, located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico. Floor to floor heights are assumed to be 14 feet.

Occupancy Live Loads:

First Floor:	Public circulation:	100 psf
Second & Third Floor:	Offices:	50 psf

Dead Loads:

Floor:	Partition 101-200 plf Carpet & Pad Metal Deck 1.5" NLWT 2.5" Steel Beams - 4.0 psf non-composite Mechanical A/C Ducts - 3.0 psf Electrical/Lighting - 2.0 psf Suspended Channel - Tile Ceiling
Roof:	Composite 5-ply/Gravel roofing Steel 1-1/2" 20 gauge deck Steel Bar Joist 24' at 4' o.c. Mechanical A/C Ducts - 3.0 psf Electrical/Lighting - 2.0 psf Rigid Roof Insulation 4" Suspended Channel - Tile Ceiling
Foundation Wall:	12" Cast-in-place Concrete

Required: Perform a wall load rundown for the foundation wall. Use the dead + live + minimum roof live load case. No live load reductions will be considered.

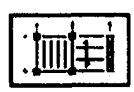
Solution:

A. Establish Criteria.

1. Re-open model from column load rundown. If unavailable, complete step A of the column load rundown example.

B. Draw Volumetric Model.

1. If 3D model is not drawn, complete step B of the column load rundown example.
2. Draw foundation wall.
   a. Select the ground floor HORIZONTAL STRUCTURAL PLANE.
   b. Select the DRAW STRUCTURE tool palette.
   c. Delete columns on the perimeter.



>> Note: We are deleting the columns which support the ground floor plane, i.e. the columns below the ground plane.

(1) Select DELETE STRUCTURE from the Edit pull-down.

(2) Click on each of the columns on the perimeter.

(3) Double click the right mouse key to end deleting structure.

d. Draw the foundation walls.

(1) Select Wall 2 GRID POINTS from the Column/Wall pull-down menu.

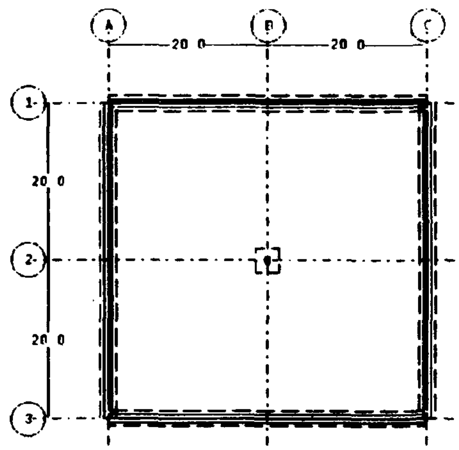
(2) Select grid location A1 and A3. A wall will appear with a continuous footing indicated below.

(3) Set the wall thickness to 12 inches within the Wall Elements dialog window.

(4) Turn OFF ALL FLOORS.

(5) Turn off ASSIGN DEAD LOAD.

(6) Select SAVE to draw the foundation wall.



(7) Repeat steps 1 through 5 for the other three walls.

>> Note: The wall dead load could have been assigned at the same time as drawing the wall. This was not done to show you the process of assigning a linear wall dead load.

>> Note: You may wish to save the model now.

C. Develop Independent Load Cases.

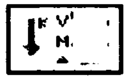
1. If dead and live floor and roof loads are not applied, complete step C of the column load rundown example.

>> Note: Save model.

2. Create and apply foundation wall dead load.

a. Select the basement floor HORIZONTAL STRUCTURAL PLANE.

» **Note: No structural elements will appear since you are viewing structure below the basement plane. The footings are connected to the walls above and thus do not appear.**



b. Prepare the wall dead load from the given load list.

(1) Select the LOADS AND DESIGN tool palette.

(2) Select WALL (DL) from the Loads pull-down menu. A Wall (DL) dialog window will appear.

(3) Select the Structure data window button and select the Concrete 8" wall load.

(4) Change the text to CONCRETE 12" and the psf to 150.

c. Turn OFF ASSIGN ALL FLOORS.

d. Assign the wall dead load between grid locations A1 to A3.

(1) Select ASSIGN.

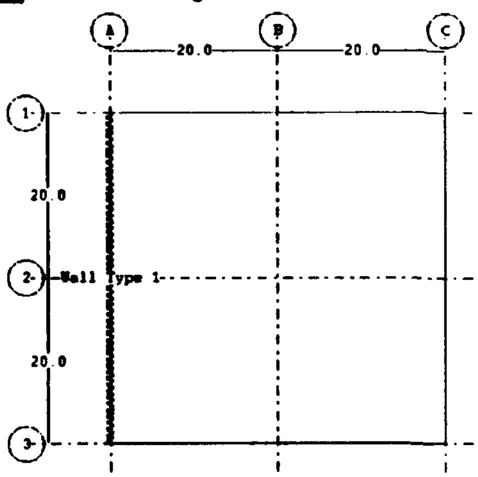
(2) Click on grid locations A1 and A3. A Wall Heights dialog window will appear.

(3) Set the wall height at each end to 14 feet.

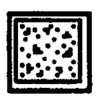
(4) Select OK to save the assigned wall dead load. The wall dead load will then appear.

» **Note: Do to symmetry, the other three wall dead loads need not be assigned.**

e. Select OK to end working with wall dead loads.



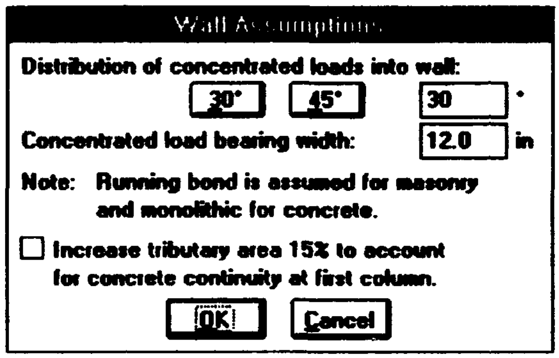
D. Establish element parameters to perform the wall load rundown for wall A1 to A3.



1. Select material: CONCRETE.
2. Select the ground floor HORIZONTAL STRUCTURAL PLANE.
3. Select component from the Column/Wall pull-down menu: WALL, C.I.P. and click on wall A1 to A3.
4. Review the data in the dialog windows and Guidelines.
5. Continue on to the Preliminary Analysis.

E. Preliminary Analysis of wall A1 to A3.

1. Select the load combination: DEAD + LIVE + MIN. ROOF.
2. Turn OFF APPLY LIVE LOAD REDUCTION in the Occupancy Live Load dialog window.
3. Select PRELIMINARY from the Design pull-down menu.
4. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L + Lr.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
5. Review Wall Assumptions dialog box.



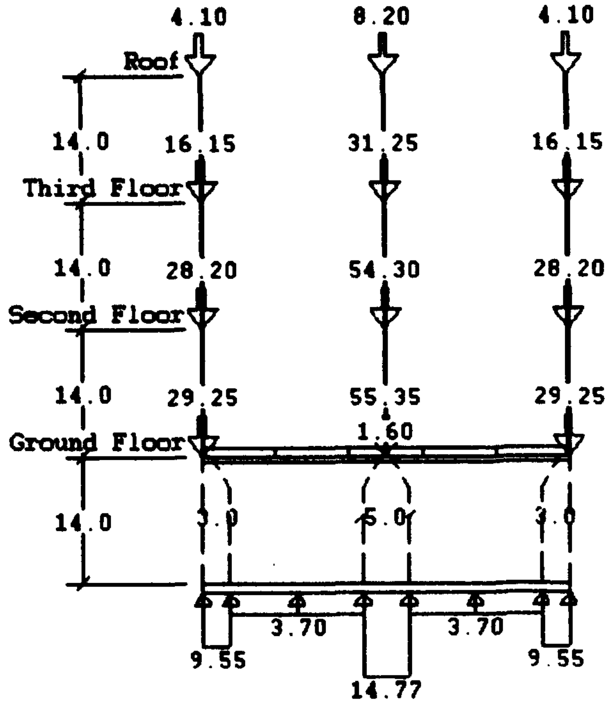
   a. Set the DISTRIBUTION OF CONCENTRATED LOADS INTO WALL to 30 degrees.

» **Note: User shall base this decision on the appropriate material and building code applicable to this project.**

   b. Set the CONCENTRATED LOAD BEARING WIDTH to 12 inches.

» **Note: User shall base this decision on estimated column size and appropriate bearing plate proportions. Pilasters are not included in this version of CASM.**

   c. Turn OFF INCREASE TRIBUTARY AREA 15% since the columns are steel.
   d. Select OK.
6. Enter a Minimum Roof Live Load calculation output file.
7. Select to use the Tributary Area Method of calculating the loads at each level.
8. Set the column self weight as 75 plf. Analysis of the wall rundown will begin.
9. Review the wall load rundowns.



Total Combined Load -- Sum (klf)

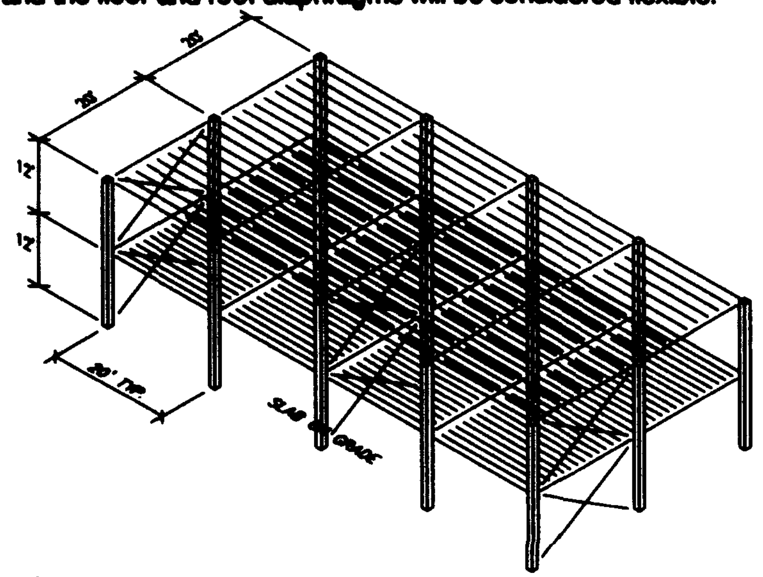
10. Select CANCEL since there is no concrete wall design spreadsheet at this time.
11. Select CANCEL in the Wall Elements dialog box.

F. Save the model as: WALL1.BLD.

■ Lateral Resistance Design

[illegible]

Given: A two story wood framed administrative building located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico as illustrated. Three equally spaced vertical lateral resistance planes will be considered and the floor and roof diaphragms will be considered flexible.



Dead Loads:

Floor:
- 1-1/8" T & G Plywood
- 5/8" Gypsum Ceiling
- 1" Carpet & Pad
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- 2 x 12 Joists @ 16" o.c.

Roof:
- 3/4" T & G Plywood
- 5/8" Gypsum Ceiling
- 5 Ply T & G Roofing
- 12" Batt Insulation
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- 2 x 12 Joists @ 24" o.c.

Required: Perform a braced frame lateral resistance analysis. Use the dead + live + min. roof LL + wind load case.

Solution:

A. Establish Criteria.

1. Open the given model LATERAL1.BLD or enter the following criteria:

Project:	Project Name	: LATERAL EXAMPLE 1
	City/Installation	: AMMO PLANT
	State	: MS
	Design Load	: TM 5-809-1 1986
Regional:	Basic Wind Speed	: 100.0 mph
	Coastal	: YES
Site:	Wind Importance	: I
	Wind Exposure	: C
	Distance to Oceanline	: 0 miles

B. Draw Volumetric Model.

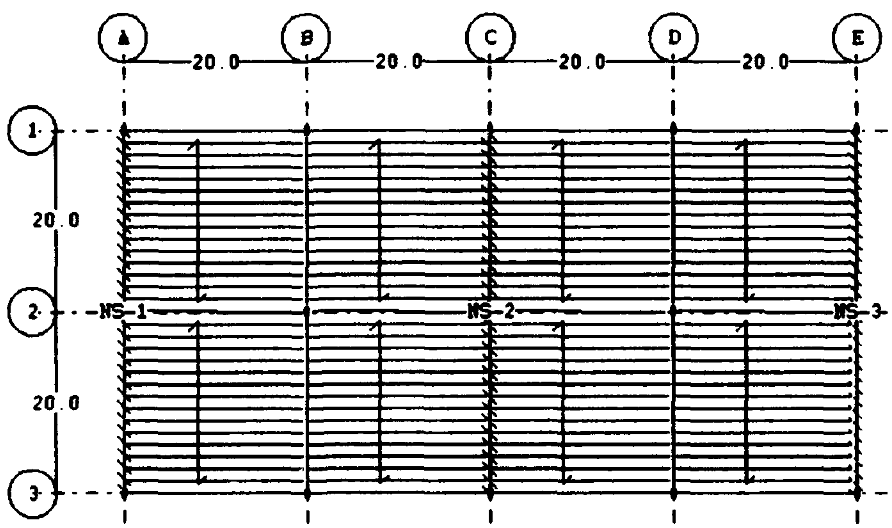
1. The 3D model is drawn in file LATERAL1.BLD or draw model from the given information.
2. Define lateral resistance vertical plane locations.


   a. Select the second floor HORIZONTAL STRUCTURAL PLANE.
   b. Select the DRAW STRUCTURE tool palette.

   c. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   d. Select beam A1 to A2. All structural elements which connect to the beam will be joined to form a vertical lateral resistance plane. Hatched lines will appear to indicate that vertical bracing will be introduced later somewhere along grid line A. The location is also labeled NS-1 to indicate lateral resistance in the north-south direction.

   e. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   f. Select beam C1 to C2.

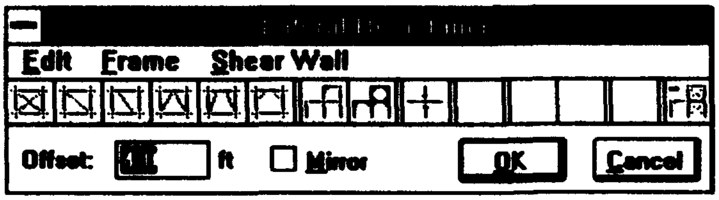
   g. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   h. Select beam E1 to E2.



3. Draw vertical bracing.
   a. Select Vertical DEFINE ELEMENTS from the Lateral pull-down menu.
   b. Select the NS-2 lateral resistance location along grid line C. A 2D elevation of the plane and a Lateral Resistance tool palette will appear.

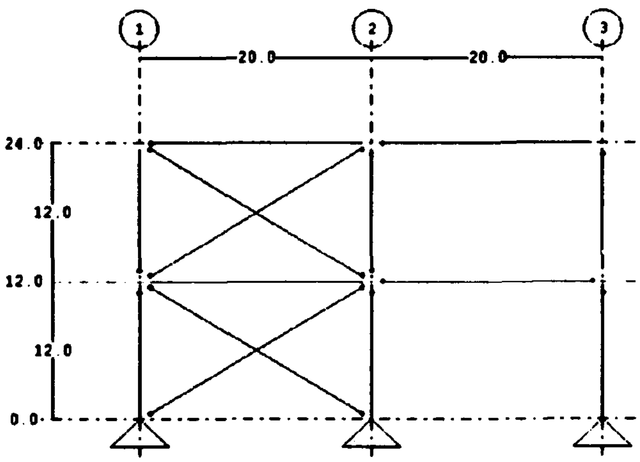
» Note:All members will initially be shown pin ended (moment = 0) and supports will be initially shown as hinges. These connection restraints can be modified as desired.

» Note:The lateral resistance tool palette includes multiple trussing options as well as rigid frame and continuous member options. The offset is measured from the column along the beam and is needed for the modified K, eccentric and knee brace options. The mirror option draws the bracing in an opposite hand position.



   c. Select the X-BRACE icon from the tool palette.

   d. Select upper left bay handle. The x-bracing will appear on the 2D view between grids 1 and 2 at the second level.
   e. Select the lower left bay handle to draw the x-bracing at the first level.



   f. Double click the right mouse key to end drawing x-bracing.
   g. Select OK when finished. You will return to the 2D horizontal structural plane.

4. Define floor and roof diaphragm type.
   a. Select DIAPHRAGM GUIDELINES to view the cardfile guidelines on selecting a rigid or flexible diaphragm.
   b. Select OK when finished viewing the diaphragm guidelines.

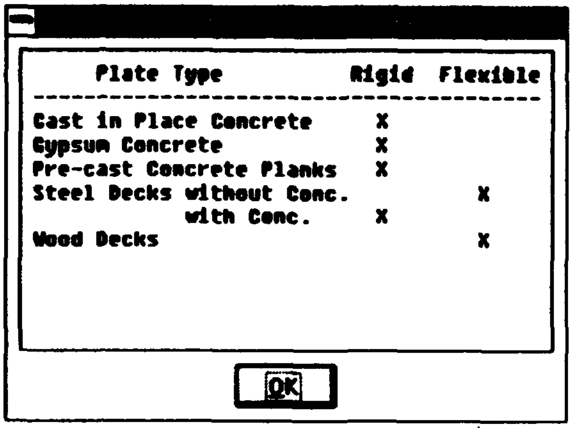


Plate Type	Rigid	Flexible
Cast in Place Concrete	X	
Gypsum Concrete	X	
Pre-cast Concrete Planks	X	
Steel Decks without Conc.		X
with Conc.	X	
Wood Decks		X

c. Select Horizontal ENTIRE PLANE FLEXIBLE DIAPHRAGM from the Lateral pull-down menu. The 2D view will be labeled as Flexible Diaphragm in the lower right corner and the flexible diaphragm icon will be highlighted in the tool palette.

>> Note: Lateral loads will be distributed to the vertical resisting planes according to tributary width or the continuous beam model at the user's choice when flexible diaphragm is selected. Lateral loads will be distributed to the vertical resisting planes according to the vertical resisting element stiffness when rigid diaphragm is selected.

>> Note: Since this is the first floor diaphragm type defined, all other floor and roof planes will also be defined as flexible.

>> Note: Only three combinations of flexible and rigid diaphragms are possible:

1. All flexible.
2. All rigid.
3. Floors rigid and roof flexible.

>> Note: You may wish to save the model now.

C. Develop Independent Load Cases.

1. The loads are already applied in file LATERAL1.BLD or apply the loads given above.

>> Note: Save model.

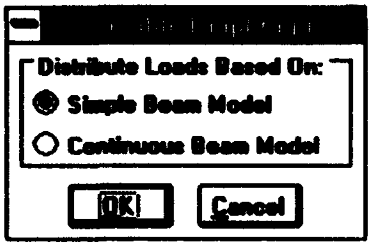
D. Establish element parameters to perform the lateral analysis.

1. Select the LOADS AND DESIGN tool palette.
2. Select material: WOOD.

E. Preliminary Lateral Analysis

1. Select the load combination: DEAD + LIVE + MIN. ROOF LL + WIND.
2. Select LATERAL RESISTANCE from the Design pull-down menu.

3. Select location NS-2 on grid line C. The 2D view of the vertical resistance plane will appear.
4. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L + Lr + W.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
5. Select the base connectivity as a hinge for all three columns.
6. Review the bracing options and keep x-bracing between grids 1 and 2.
7. Select the Wind Direction as SOUTH and the Wind Load as GCPI= 0.
8. Enter a Minimum Roof Live Load output filename.
9. Select SIMPLE BEAM MODEL from the Flexible Diaphragm dialog window to distribute the wind loads according to tributary width.

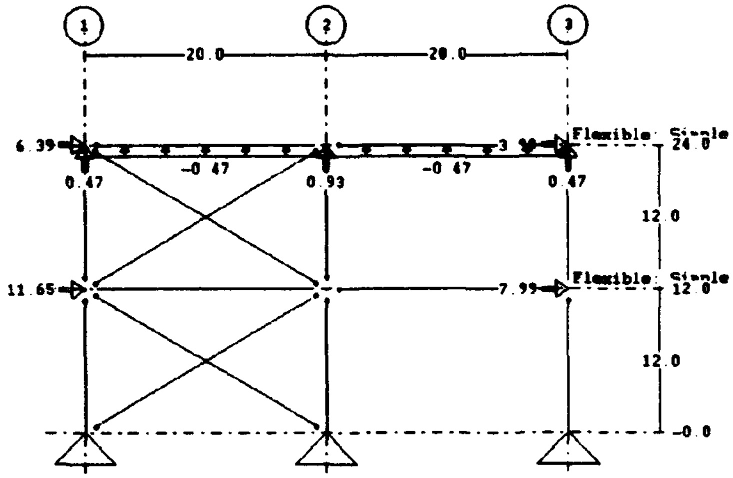


10. Review the loads on the braced frame.

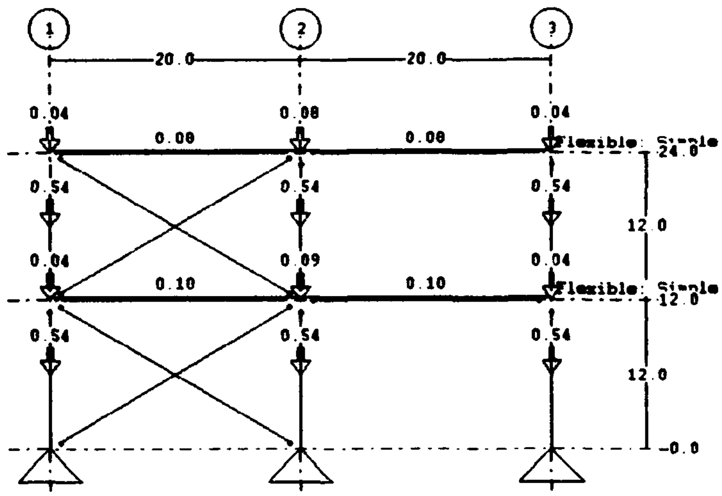
» Note: The wind loads acting upward indicate roof suction.

11. Enter the self weight of all beams as 36 plf and turn ON ADD SELF WEIGHT.
12. Enter the self weight of all columns as 45 plf.
13. Review the loads on the braced frame.

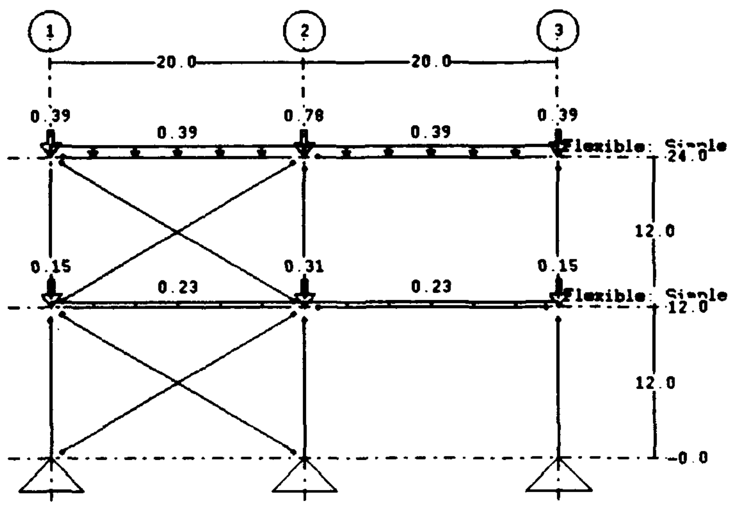
» Note:The column self weight is treated as a concentrated load at the mid-height of each column.



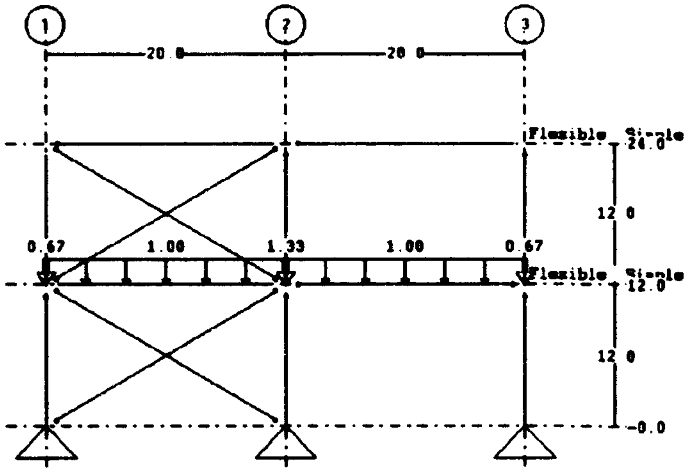
1.00 Wind (klf)



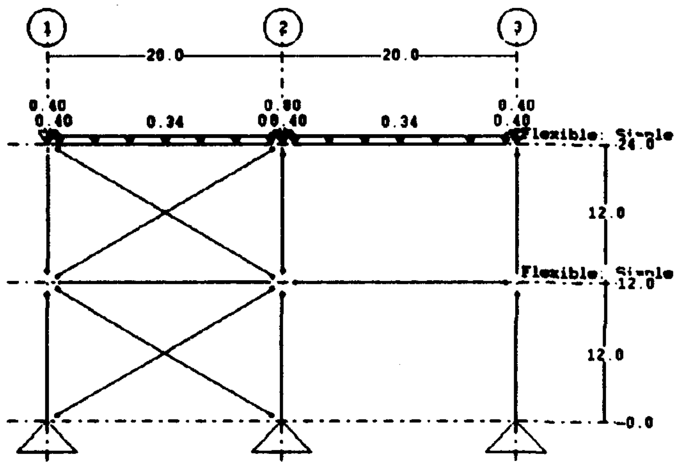
1.00 Dead (klf)



1.00 Superimposed Dead (klf)



1.00 Live (klf)



1.00 Minimum Roof (klf)

14. Enter an appropriate analysis file name.

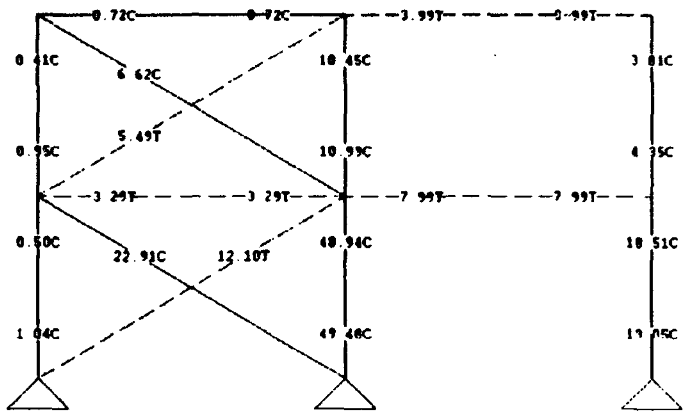
>> Note: Compression members are not automatically removed in the analysis.

>> Note: Each member is divided into four segments for the purpose of plotting shear, moment and deflection diagrams.

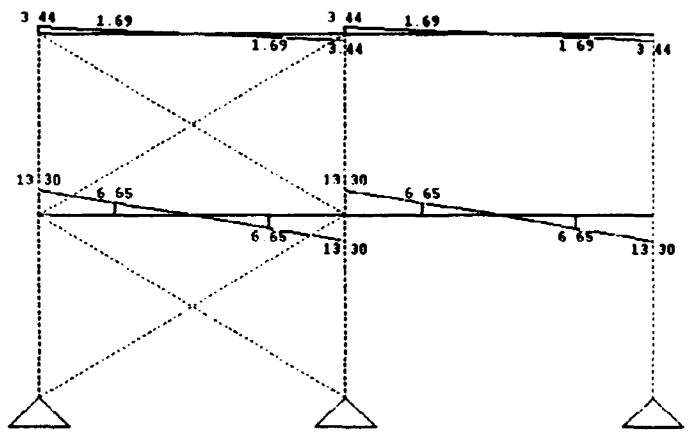
>> Note: Large lateral load structural models require a significant amount of memory to perform the analysis, so an out of memory error could occur.

15. Review the axial, shear, moment, deflection, and loads and reactions diagrams.

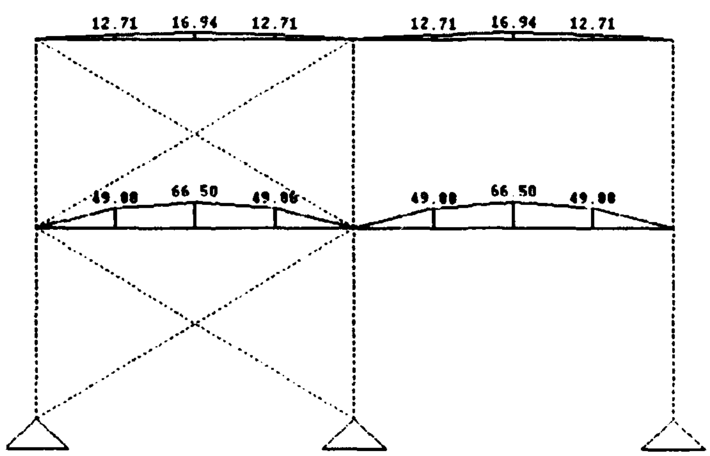
>> Note: Low-rise braced frames experience small lateral deflections. Therefore, increase the X Deflection Scale to 300. Review hard copy analysis output for relative deflection magnitudes.



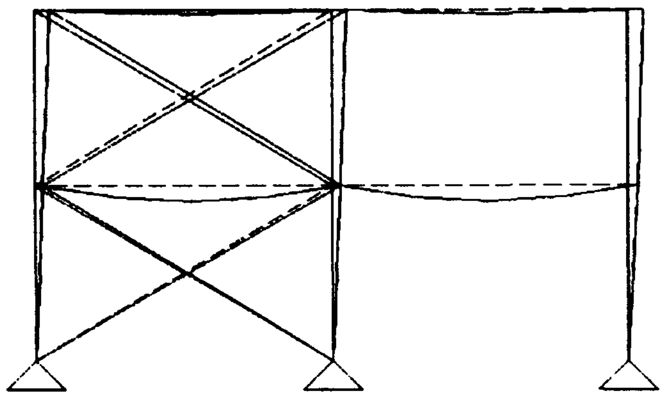
Total Combined Load -- Axial (k)



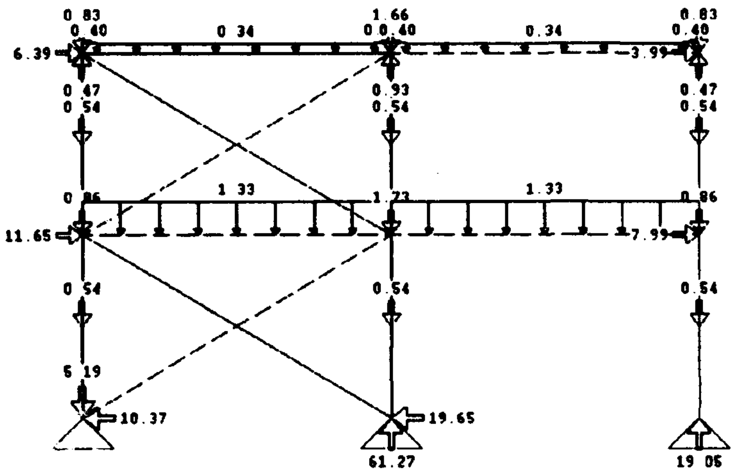
Total Combined Load — Shear (k)



Total Combined Load -- Moment (kft)



Total Combined Load — Deflection



Total Combined Load -- Loads & Reactions (k)

16. Select CANCEL since there is no lateral resistance member design at this time.

F. Save the model as: LAT1.BLD.

[illegible]

Given: The same two story wood framed administrative building located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico as illustrated in lateral example 1. Three equally spaced vertical lateral resistance planes will be considered and the floor and roof diaphragms will be considered flexible.

Dead Loads:

Floor:	1-1/8" T & G Plywood
	5/8" Gypsum Ceiling
	1" Carpet & Pad
	Mechanical - 3.0 psf
	Electrical - 1.0 psf
	2 x 12 Joists @ 16" o.c.
Roof:	3/4" T & G Plywood
	5/8" Gypsum Ceiling
	5 Ply T & G Roofing
	12" Batt Insulation
	Mechanical - 3.0 psf
	Electrical - 1.0 psf
	2 x 12 Joists @ 24" o.c.

Required: Perform an unbraced frame lateral resistance analysis. Use the dead + live + min. roof LL + wind load case.

Solution:

A. Establish Criteria.

1. Open the saved model LAT1.BLD or enter the following criteria:

Project:	Project Name	: LATERAL EXAMPLE 2
	City/Installation	: AMMO PLANT
	State	: MS
	Design Load	: TM 5-809-1 1986
Regional:	Basic Wind Speed	: 100.0 mph
	Coastal	: YES
Site:	Wind Importance	: I
	Wind Exposure	: C
	Distance to Oceanline	: 0 miles

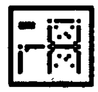
2. If using the saved example, change the Project Name to LATERAL EXAMPLE 2.

B. Draw Volumetric Model.

1. The 3D model is drawn in file LAT1.BLD or draw model from the given information.
2. Define lateral resistance vertical plane locations. We will use the same lateral resistance plane locations as in lateral example 1.
3. Draw vertical frames.

a. Select the second floor HORIZONTAL STRUCTURAL PLANE.
b. Select the DRAW STRUCTURE tool palette.
c. Select Vertical DEFINE ELEMENTS from the Lateral pull-down menu.

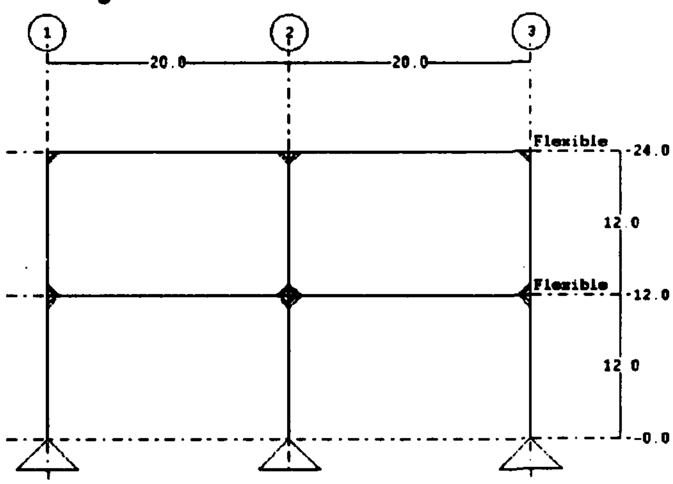
d. Select the NS-2 lateral resistance location along grid line C. A 2D elevation of the plane and a Lateral Resistance tool palette will appear.
e. Delete the x-bracing from the lateral resistance plane.



(1) Select the DELETE ELEMENT button. Handles will appear on the x-bracing.
(2) Select one of the bracing's handles then the other one.

>> Note: The delete lateral resistance elements command automatically ends when there are no more elements to delete. Otherwise, to end the delete lateral resistance elements command, double click the right mouse key.

f. Select the ALL MOMENT CONNECTIONS icon from the tool palette. A full continuity moment connection is graphically illustrated by a double lined triangle.



g. Select OK when finished. You will return to the 2D horizontal structural plane.

4. Define floor and roof diaphragm type.

a. Flexible floor and roof diaphragms are already defined.

>> Note: You may wish to save the model now.




C. **Develop Independent Load Cases.**

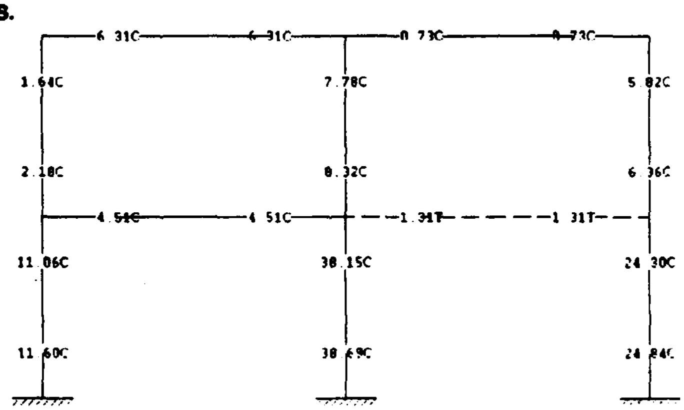
1. The loads are already applied in file LAT1.BLD or apply the loads given above.

D. **Establish element parameters to perform the lateral analysis.**

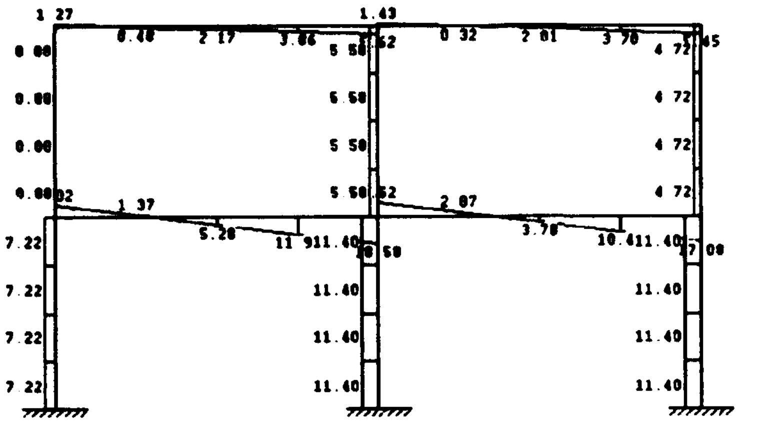
1. Select the LOADS AND DESIGN tool palette.
2. Select material: WOOD.

E. **Preliminary Lateral Analysis**

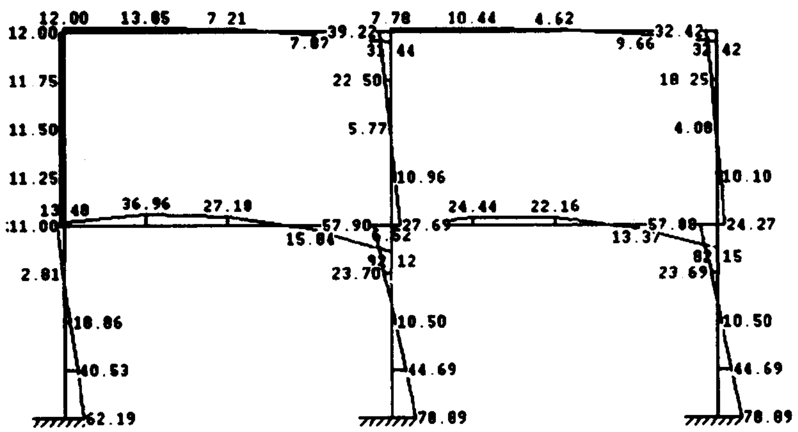
1. Select the load combination: DEAD + LIVE + MIN. ROOF LL + WIND.
2. Select LATERAL RESISTANCE from the Design pull-down menu.
3. Select location NS-2 on grid line C. The 2D view of the vertical resistance plane will appear.
4. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L + Lr + W.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL= DECK+ SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
5. Change the base connectivity to full fixity for all three columns.
6. Review the bracing options and keep the rigid moment connections between all members.
7. Select the Wind Direction as SOUTH and the Wind Load as GCPI= 0.
8. Enter a Minimum Roof Live Load output filename.
9. Select SIMPLE BEAM MODEL in the Flexible Diaphragm dialog window.
10. Review the loads on the unbraced frame.
11. Enter the self weight of all beams as 36 plf and turn ON ADD SELF WEIGHT.
12. Enter the self weight of all columns as 45 plf.
13. Review the loads on the unbraced frame.
14. Enter an appropriate analysis file name.
15. Review the axial, shear, moment, deflection, and loads and reactions diagrams.



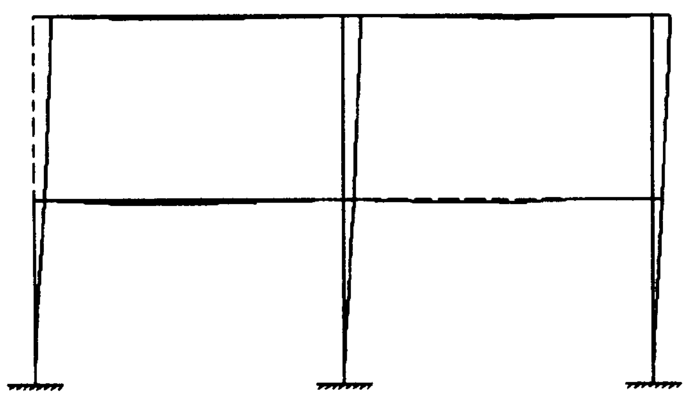
Total Combined Load -- Axial (k)



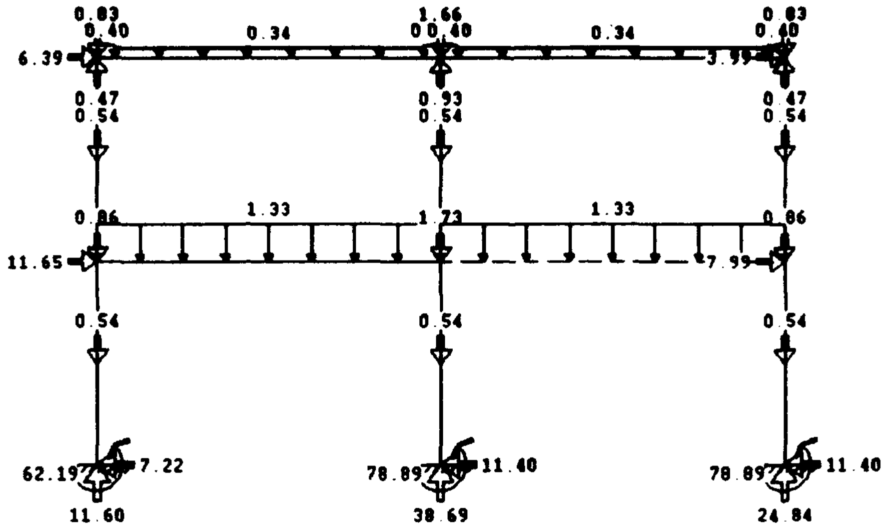
Total Combined Load -- Shear (k)



Total Combined Load — Moment (kft)



Total Combined Load -- Deflection

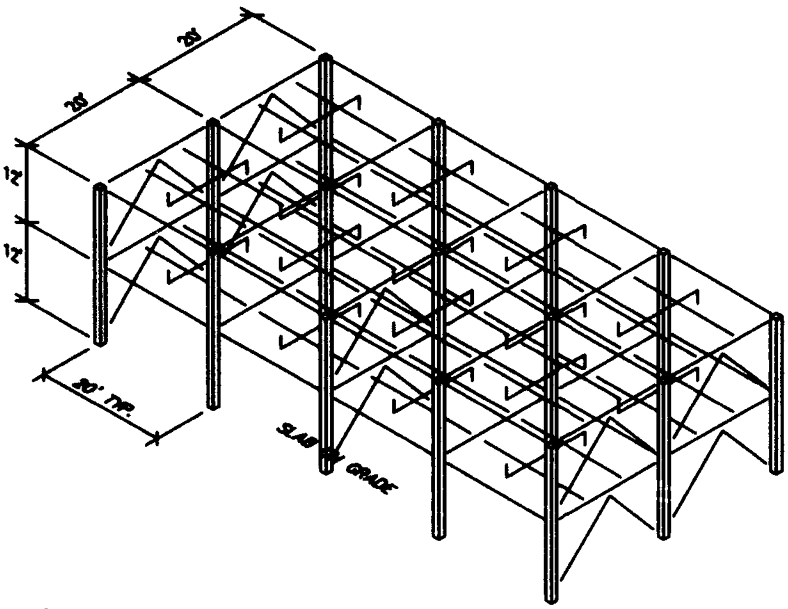


Total Combined Load — Loads & Reactions (k)

16. Select CANCEL since there is no lateral resistance member design at this time.

F. Save the model as: LAT2.BLD.

Given: A two story steel framed administrative building located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico as illustrated. Three equally spaced vertical lateral resistance planes will be considered and the floor and roof diaphragms will be considered rigid. This example will utilize trussing elements for lateral resistance.



Dead Loads:

Floor:
- 1-1/2" + 2-1/2" NLWT Concrete Deck
- Suspended Ceiling
- 1" Carpet & Pad
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- Steel Beams - 3.3 psf

Roof:
- 1-1/2" + 2-1/2" NLWT Concrete Deck
- Suspended Ceiling
- 5 Ply Built-up Roofing
- 4" Rigid Insulation
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- Steel Beams - 3.3 psf

Required: Perform a braced frame lateral resistance analysis. Use the dead + live + min. roof LL + wind load case.

Solution:

A. Establish Criteria.

1. Open the given model LATERAL3.BLD or enter the following criteria:

Project:	Project Name	: LATERAL EXAMPLE 3
	City/Installation	: AMMO PLANT
	State	: MS
	Design Load	: TM 5-809-1 1986
Regional:	Basic Wind Speed	: 100.0 mph
	Coastal	: YES
Site:	Wind Importance	: I
	Wind Exposure	: C
	Distance to Oceanline	: 0 miles

B. Draw Volumetric Model.

1. The 3D model is drawn in file LATERAL3.BLD or draw model from the given information.
2. Define lateral resistance vertical plane locations.


a. Select the second floor HORIZONTAL STRUCTURAL PLANE.

b. Select the DRAW STRUCTURE tool palette.

c. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.

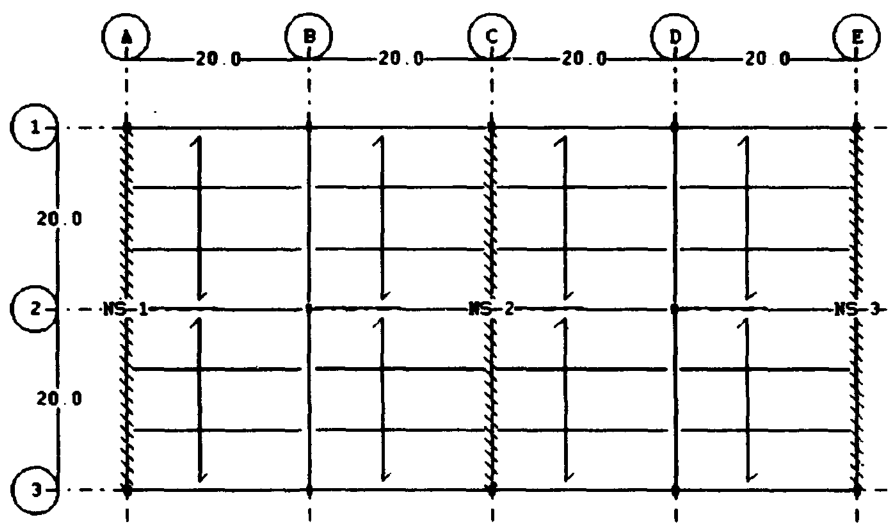
d. Select beam A1 to A2. All structural elements which connect to the beam will be joined to form a vertical lateral resistance plane. Hatched lines will appear to indicate that vertical bracing will be introduced later somewhere along grid line A. The location is also labeled NS-1 to indicate lateral resistance in the north-south direction.

e. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.

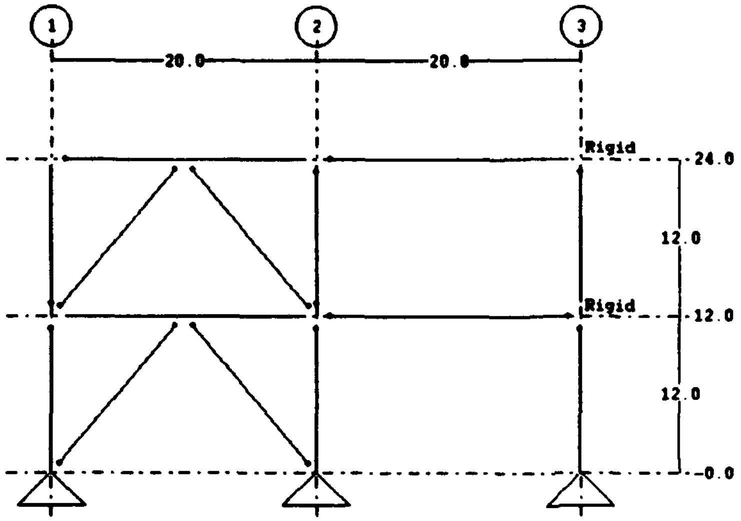
f. Select beam C1 to C2.

g. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.

h. Select beam E1 to E2.



3. Draw vertical bracing.
   a. Select Vertical DEFINE ELEMENTS from the Lateral pull-down menu.
   b. Select the NS-2 lateral resistance location along grid line C. A 2D elevation of the plane and a Lateral Resistance tool palette will appear.
   c. Select the K-BRACE icon from the tool palette.
   d. Select upper left bay handle. The k-bracing will appear on the 2D view between grids 1 and 2 at the second level.
   e. Select the lower left bay handle to draw the k-bracing at the first level.
   f. Double click the right mouse key to end drawing k-bracing.



   g. Select OK when finished. You will return to the 2D horizontal structural plane.

» Note: For rigid diaphragms, all frames in one direction are analyzed with a one kip load applied at the top element to compute the distribution of wind load based on stiffness. Therefore, each lateral resistance location must have bracing elements or moment connection elements defined.

   h. Draw k-bracing elements in all four panels in lateral resistance locations NS-1 and NS-3 by following steps a through f.

4. Define floor and roof diaphragm type.
   a. Select Horizontal RIGID DIAPHRAGM from the Lateral pull-down menu. The 2D view will be labeled as Rigid Diaphragm in the lower right corner and the rigid diaphragm icon will be highlighted in the tool palette.

» Note:Lateral loads will be distributed to the vertical resisting planes according to the vertical resisting element stiffness when a rigid diaphragm is selected. Lateral loads will be distributed to the vertical resisting planes according to tributary width or the continuous beam model when a flexible diaphragm is selected.

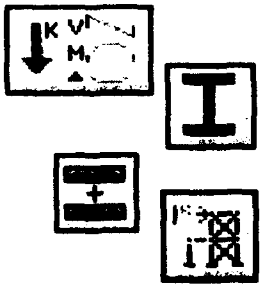
» Note:Since this is the first floor diaphragm type defined, all other floors and roof planes will also be defined as rigid.

» Note:You may wish to save the model now.

C. Develop Independent Load Cases.

1. The loads are already applied in file LATERAL3.BLD or apply the loads given above.

» Note: Save model.

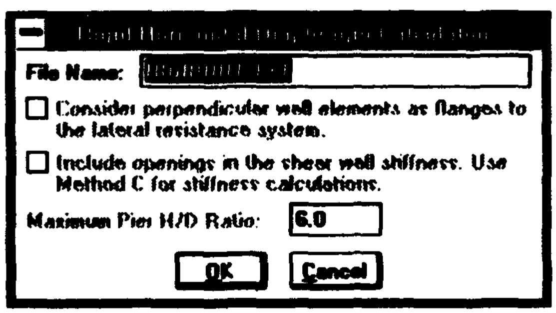


D. Establish element parameters to perform the lateral analysis.

1. Select the LOADS AND DESIGN tool palette.
2. Select material: STEEL.

E. Preliminary Lateral Analysis

1. Select the load combination: DEAD + LIVE + MIN. ROOF LL + WIND.
2. Select LATERAL RESISTANCE from the Design pull-down menu.
3. Select location NS-2 on grid line C. The 2D view of the vertical resistance plane will appear.
4. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L + Lr + W.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL=DECK+SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
5. Select the base connectivity as a hinge for all three columns.
6. Review the bracing options and keep k-bracing between grids 1 and 2.
7. Select the Wind Direction as SOUTH and the Wind Load as GCPI=0. Each lateral resistance plane is analyzed for a one kip lateral force to compare stiffness for wind load distribution.
8. Enter a Minimum Roof Live Load output filename.
9. Enter an appropriate filename for the rigid horizontal diaphragm calculations.



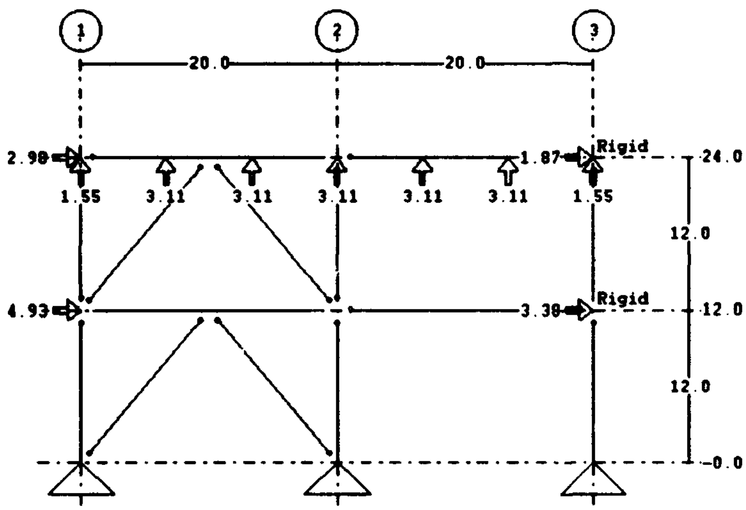
» NOTE: The options to consider perpendicular wall elements as flanges and including openings only pertain to shear walls.

10. Review the loads on the braced frame.

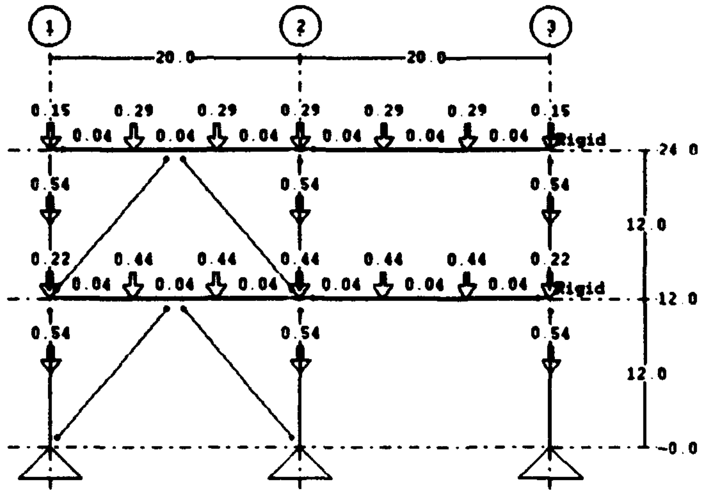
» Note: The percentage of wind load distributed by stiffness at each level is given. The outer braced frames receive a greater percentage of wind load since they had a greater stiffness than the center braced frame. If all three braced frames had equal stiffness, the load distribution would be 33.3% to each one. Had the diaphragms been flexible, the load distribution would have been 25% to each end braced frame and 50% to the middle braced frame.

» Note:Analysis can only be performed on the selected middle lateral resistance plane even though it is possible to review the other lateral resistance planes' wind loads.

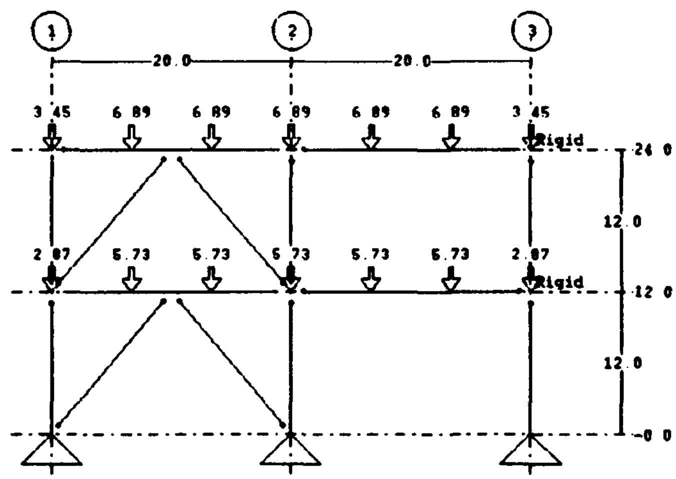
11. Enter the self weight of all beams as 36 plf and turn ON ADD SELF WEIGHT.
12. Enter the self weight of all columns as 45 plf.
13. Review the loads on the braced frame.



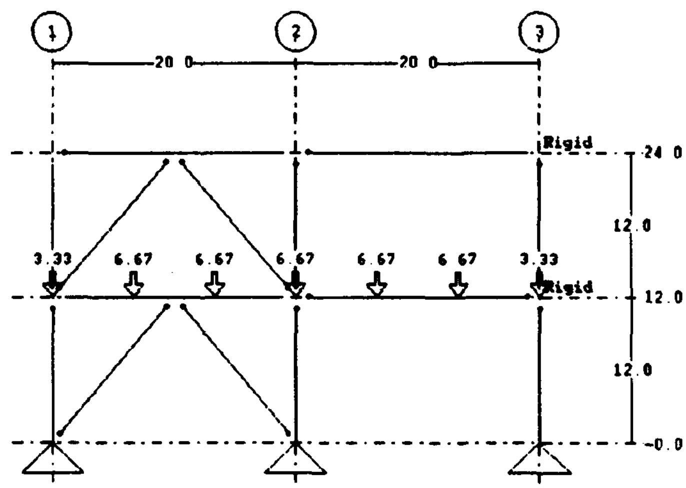
1.00 Wind (klf) — NS-2 — 23%, 21%



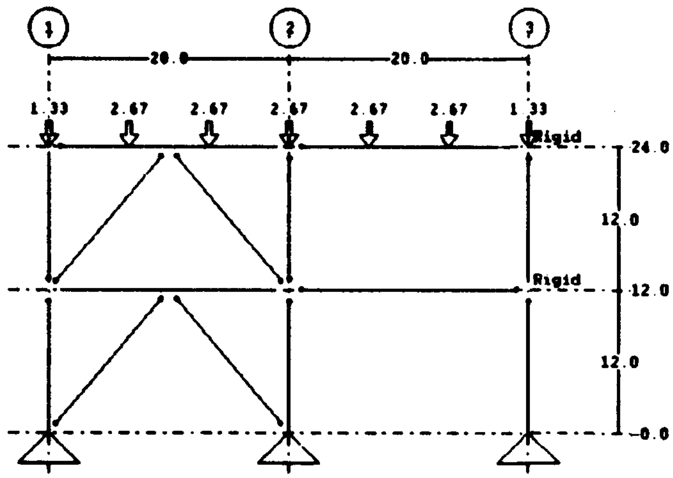
1.00 Dead (klf)



1.00 Superimposed Dead (klf)

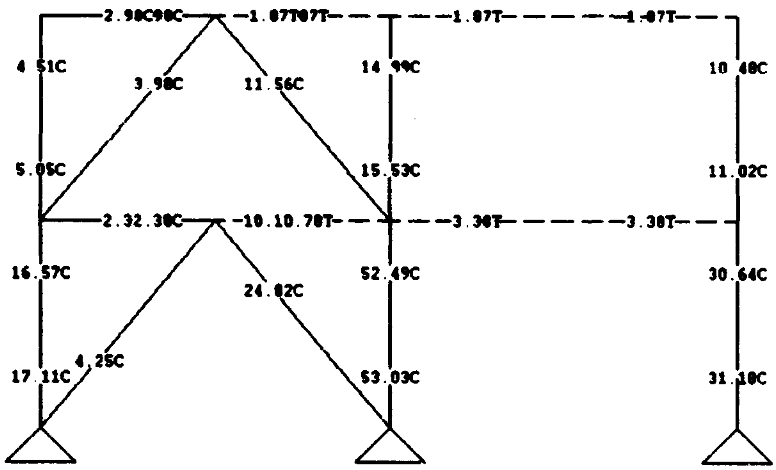


1.00 Live (klf)

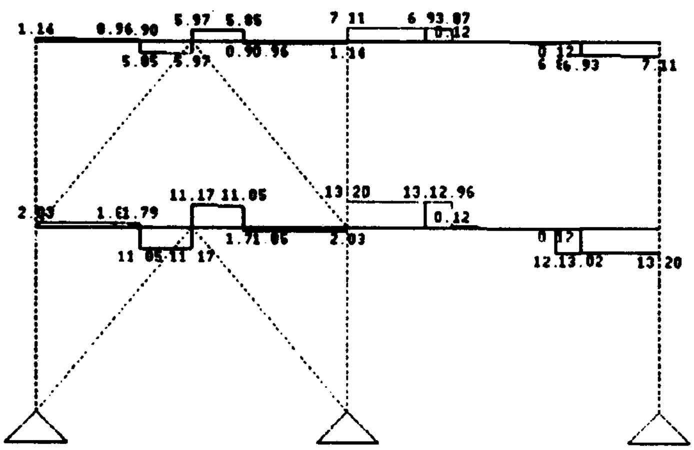


1.00 Minimum Roof (klf)

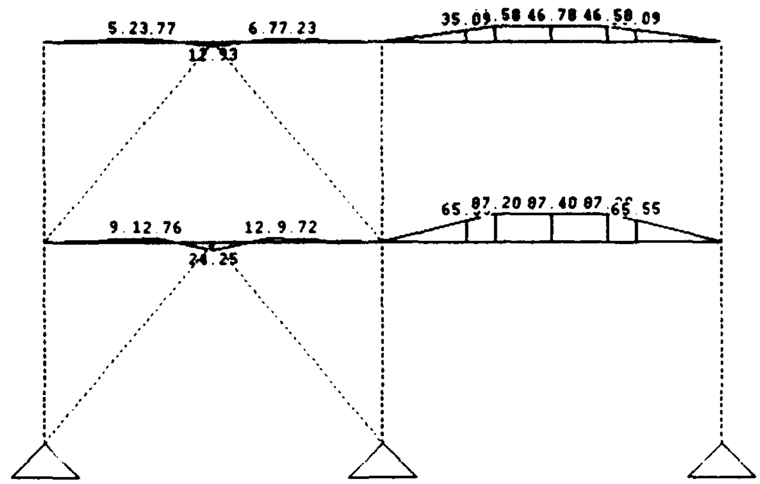
14. **Enter an appropriate analysis file name.**
15. **Review the axial, shear, moment, deflection, and loads and reactions diagrams.**



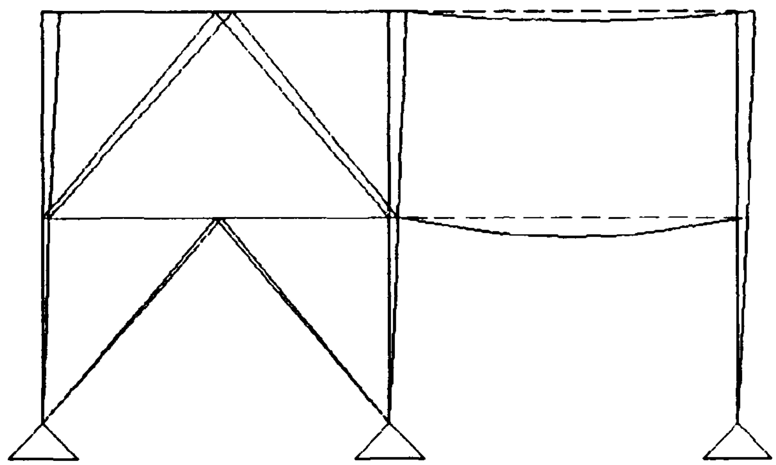
Total Combined Load — Axial (k)



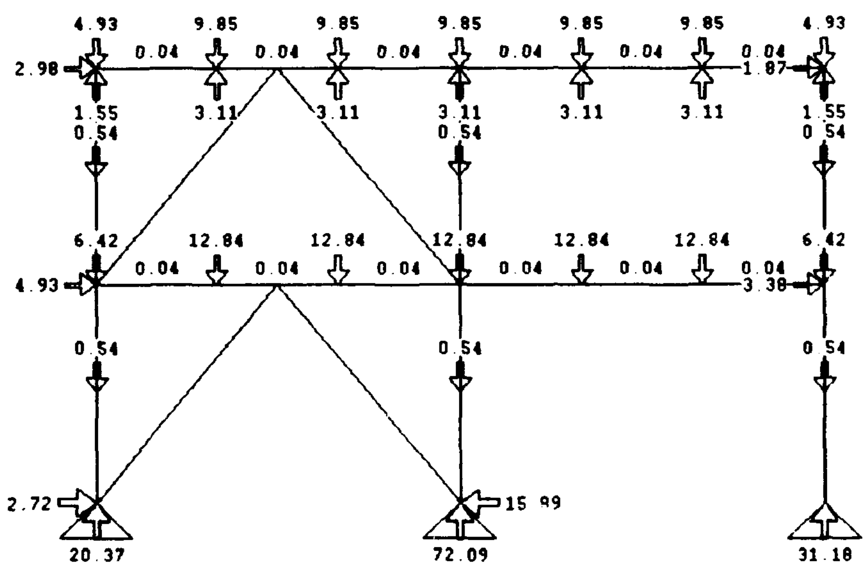
Total Combined Load — Shear (k)



Total Combined Load -- Moment (kft)



Total Combined Load -- Deflection



Total Combined Load -- Loads & Reactions (k)

16. **Select CANCEL since there is no lateral resistance member design at this time.**
17. **Review the Rigid Horizontal Diaphragm Calculation.**
    a. **Select PRINT DATA from the File pull-down menu.**
    b. **Select only the RIGID DIAPHRAGM output file.**
    c. **Select either PRINT TO PRINTER or PRINT TO FILE and select OK.**
    d. **Review the center of rigidity calculations in the north-south direction. When you are finished reviewing the output, return back to CASM.**

>> Note: The assumptions used in the calculation of interstory deflections are included in the output.

>> Note: The equations for torsional moment and distribution of torsional shear are included in the output with the Fv+Ft magnitudes shown on the elevations previously viewed.

Example 7 Sample Output:

Project : Lateral Example 3
Location : Ammo Plant
Time : Wed Sep 25, 1991 12:27 PM

***************** Rigid Horizontal Diaphragm Calculations *****************

Center of Rigidity

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
NS-1	12.0	0	0	155.103	0.006	39.41%	0.0	0.000
NS-2	12.0	0	0	288.684	0.003	21.18%	40.0	0.139
NS-3	12.0	0	0	155.103	0.006	39.41%	80.0	0.516
Sum					0.016			0.654

Centroid from lower left = sum(R*x)/sum(R) : 40.00 ft
Maximum dimension : 80.00 ft
Eccentricity (e) = centroid-(max dimension)/2 : 0.00 ft
e min = 0.05*max. dimension : 4.00 ft
Eccentricity (e) used for torsional analysis : 0.00 ft
e min considered only for seismic analysis.

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
NS-1	24.0	0	0	451.601	0.002	38.32%	0.0	0.000
NS-2	24.0	0	0	741.048	0.001	23.35%	40.0	0.054
NS-3	24.0	0	0	451.601	0.002	38.32%	80.0	0.177
Sum					0.006			0.231

Centroid from lower left = sum(R*x)/sum(R) : 40.00 ft
Maximum dimension : 80.00 ft
Eccentricity (e) = centroid-(max dimension)/2 : 0.00 ft
e min = 0.05*max. dimension : 4.00 ft
Eccentricity (e) used for torsional analysis : 0.00 ft
e min considered only for seismic analysis.

Assumptions used:
Deflections calculated by applying a 1 kip load.

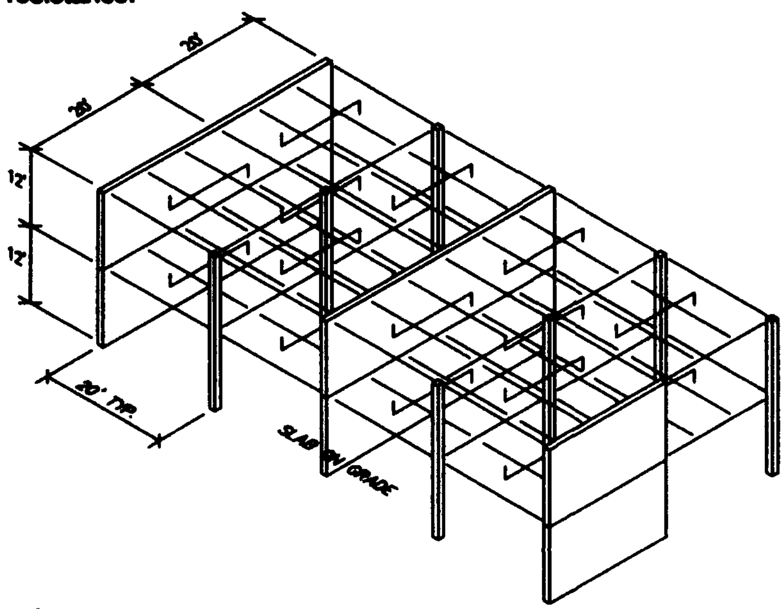
Name	h (ft)	Rigidity	dx (ft)	R*dx	R*dx*dx	R*dx/ sum(R*dx*dx)
NS-1	12.0	0.006	40.0	0.258	10.316	0.01250
NS-2	12.0	0.003	0.0	0.000	0.000	0.00000
NS-3	12.0	0.006	40.0	0.258	10.316	0.01250
Sum					20.631	

Name	h (ft)	Rigidity	dx (ft)	R*dx	R*dx*dx	R*dx/ sum(R*dx*dx)
NS-1	24.0	0.002	40.0	0.089	3.543	0.01250
NS-2	24.0	0.001	0.0	0.000	0.000	0.00000
NS-3	24.0	0.002	40.0	0.089	3.543	0.01250
Sum					7.086	

```
Shear distribution     : Fv = V*R/sum(R)
Torsional moment       : Mt = V*e
Torsional component    : Ft = Mt*R*dx/sum(R*dx*dx)
Total shear to element: Ftotal = Fv + Ft
```

F. Save the model as: LAT3.BLD.

Given: The same two story steel framed administrative building located in Mississippi at the Army Ammunition Plant on the coast of the Gulf of Mexico as shown in lateral example 3. Three equally spaced vertical lateral resistance planes will be considered and the floor and roof diaphragms will be considered rigid. This example will utilize masonry shear walls for lateral resistance.



Dead Loads:

Floor:
- 1-1/2" + 2-1/2" NLWT Concrete Deck
- Suspended Ceiling
- 1" Carpet & Pad
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- Steel Beams - 3.3 psf

Roof:
- 1-1/2" + 2-1/2" NLWT Concrete Deck
- Suspended Ceiling
- 5 Ply Built-up Roofing
- 4" Rigid Insulation
- Mechanical - 3.0 psf
- Electrical - 1.0 psf
- Steel Beams - 3.3 psf

Required: Perform a shear wall lateral resistance analysis in the N-S direction only. Use the dead + live + min. roof LL + wind load case.

Solution:

A. Establish Criteria.

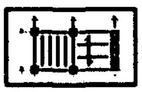
1. Open the given model LATERAL4.BLD or enter the following criteria:

Project:	Project Name	: LATERAL EXAMPLE 4
	City/Installation	: AMMO PLANT
	State	: MS
	Design Load	: TM 5-809-1 1986
Regional:	Basic Wind Speed	: 100.0 mph
	Coastal	: YES
Site:	Wind Importance	: I
	Wind Exposure	: C
	Distance to Oceanline	: 0 miles

B. Draw Volumetric Model.

1. The 3D model is drawn in file LATERAL4.BLD or draw model from the given information.

» **Note:It is necessary to draw the structural walls to be eventually considered as shear walls for lateral resistance under 'Draw Structure'. This should not be confused with planes drawn under 'Draw Model' as walls.**

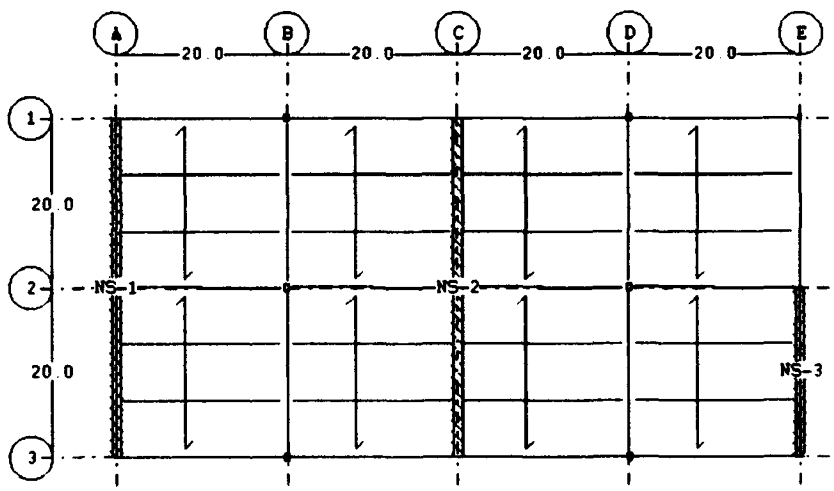


2. Define lateral resistance vertical plane locations.
   a. Select the second floor HORIZONTAL STRUCTURAL PLANE.
   b. Select the DRAW STRUCTURE tool palette.

   c. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   d. Select wall on grid line A. All in plane structural walls above and below which connect to the wall will be joined to form a vertical lateral resistance plane. Hatched lines will appear to indicate the vertical bracing location. The location is also labeled NS-1 to indicate lateral resistance in the north-south direction.

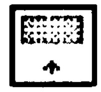
   e. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   f. Select wall on grid line C.

   g. Select Vertical DEFINE LOCATION from the Lateral pull-down menu.
   h. Select wall on grid line E.



3. Define floor and roof diaphragm type.

   a. Select Horizontal RIGID DIAPHRAGM from the Lateral pull-down menu. The 2D view will be labeled as Rigid Diaphragm in the lower right corner and the rigid diaphragm icon will be highlighted in the tool palette.



» Note: Lateral loads will be distributed to the vertical resisting planes according to the vertical resisting element stiffness when rigid diaphragm is selected. Lateral loads will be distributed to the vertical resisting planes according to tributary width or based on a continuous beam model at the user's choice when flexible diaphragm is selected.

» Note: Since this is the first floor diaphragm type defined, all other floors and roof planes will also be defined as rigid.

» Note:You may wish to save the model now.

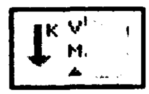
C. Develop Independent Load Cases.

   1. The loads are already applied in file LATERAL4.BLD or apply the loads given above.

» Note: Save model.

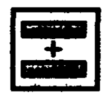
D. Establish element parameters to perform the lateral analysis.

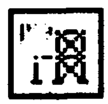
   1. Select the LOADS AND DESIGN tool palette.
   2. Select material: MASONRY.



E. Preliminary Lateral Analysis

   1. Select the load combination: DEAD + LIVE + MIN. ROOF LL + WIND.
   2. Select LATERAL RESISTANCE from the Design pull-down menu.
   3. Select shear wall location NS-2 on grid line C. The 2D view of the vertical resistance plane will appear.





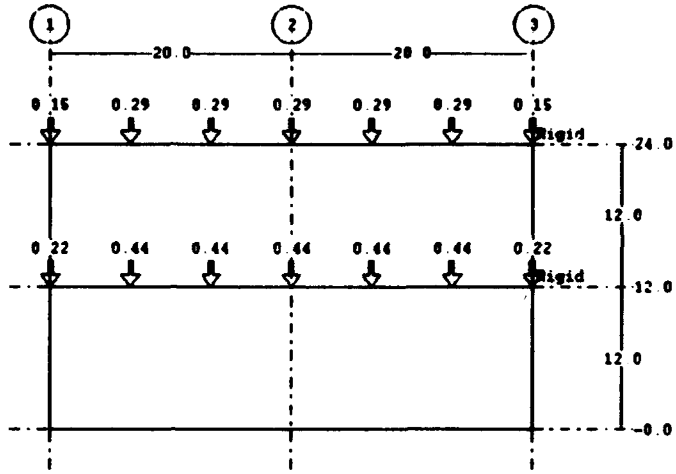
   4. Select Units options:
      a. Select units of FEET and KIPS.
      b. Verify load combination of D + L + Lr + W.
      c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, or DL=DECK+SELF WEIGHT.
   5. Select the Wind Direction as SOUTH and the Wind Load as GCPI=0. Each lateral resistance plane is analyzed for a one thousand kip lateral force to compare stiffness for wind load distribution.

» Note: A one thousand kip force is used to compare shear wall stiffnesses. A one kip force is used to compare rigid frame stiffnesses. A one kip force is used to compare trussed bracing stiffnesses.

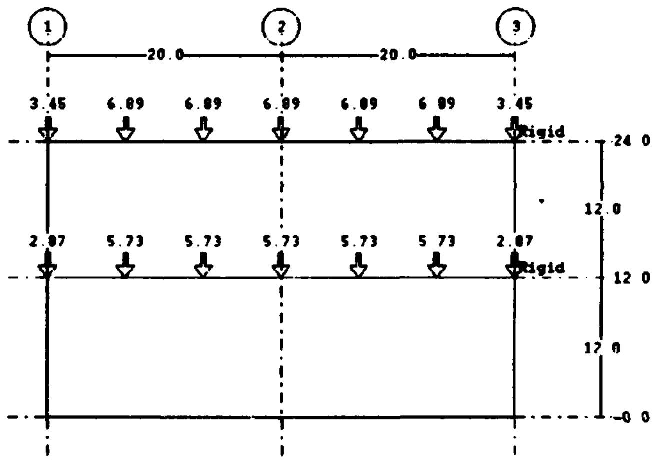
   6. Enter a Minimum Roof Live Load output filename.
   7. Enter an appropriate filename for the rigid horizontal diaphragm calculations, turn OFF CONSIDER PERPENDICULAR WALL ELEMENTS AS FLANGES TO THE LATERAL RESISTANCE SYSTEM and turn off IN-

<u>CLUDE OPENINGS IN THE SHEAR WALL STIFFNESS</u>. The load calculations will begin.

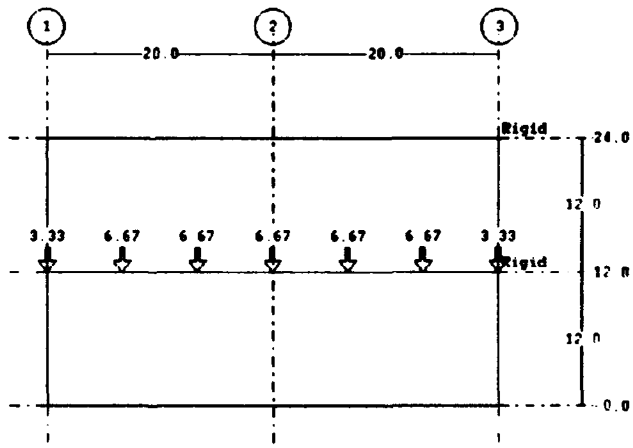
8. Review the loads on the shear wall.



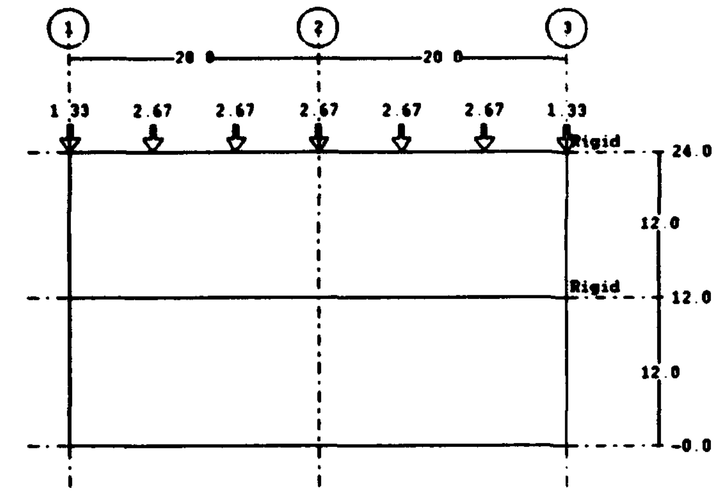
1.00 Dead (klf)



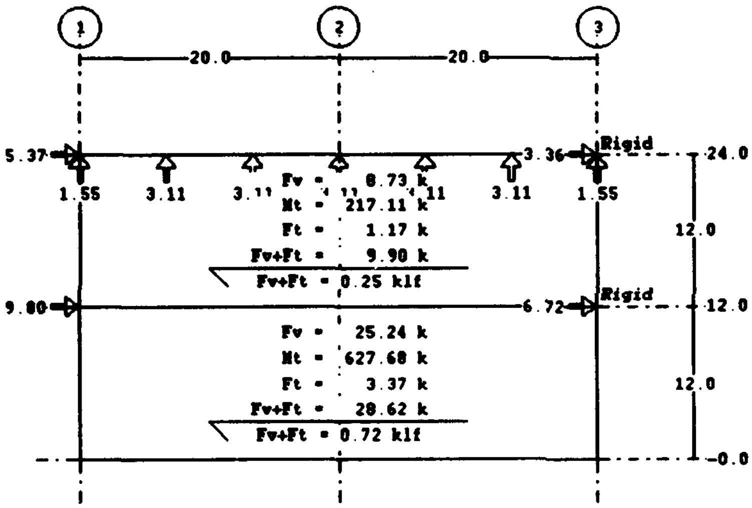
1.00 Superimposed Dead (klf)



1.00 Live (klf)



1.00 Minimum Roof (klf)



1.00 Wind (klf) — NS-2 — 42%, 42%

>> **Note: The self weight of the shear wall is not included in the dead load diagram.**

>> **Note: This version of CASM does not perform an analysis of the shear wall.**

9. **Select CANCEL since there is no lateral resistance member design at this time.**
10. **Review the Rigid Horizontal Diaphragm Calculations.**
    a. **Select PRINT DATA from the File pull-down menu.**
    b. **Select only the RIGID DIAPHRAGM output file.**
    c. **Select either PRINT TO PRINTER or PRINT TO FILE and select OK.**
    d. **Review the cross sectional properties for each resisting element taken about the north-south and east-west centroidal axes.**

» **Note: The stiffness of resisting elements are computed at each level.**

» **Note: The cross sectional properties of resisting elements that occur repetitively are not duplicated in the output file.**

e. **Review the center of rigidity calculations in the north-south direction. When you are finished, return to CASM.**

» **Note: The assumptions used in the calculation of interstory deflection are included in the output.**

» **Note: The equations for torsional moment and distribution of torsional shear are included in the output with the magnitudes shown on the wall elevations previously viewed.**

Example 8 Sample Output:

```
Project       : Lateral Example 4
Location      : Ammo Plant
Time          : Wed Sep 25, 1991   2:00 PM

***************** Rigid Horizontal Diaphragm Calculations *****************

---------------------------------------------------------------------------
                                   NS-1
---------------------------------------------------------------------------

                          Level Height: 12.0 ft
---------------------------------------------------------------------------
                              Centroidal Axis
                                           NS    NS Moment     EW   EW Moment
Name      t        l          Area        Arm       Area       Arm      Area
         (ft)     (ft)       (ft^2)      (ft)     (ft^3)      (ft)    (ft^3)
---------------------------------------------------------------------------
NS-1     1.00    40.00         40.0     20.00        800      0.00         0
---------------------------------------------------------------------------
Sum                            40.0                  800                   0

Centroid = sum(MomentArea)/sum(Area)
NS Centroid       :   20.00 ft   EW Centroid       :    0.00 ft
Av                :   40.00 sqft

                             Moment of Inertia
                           bh^3/
Name      b        h        12          Area        d          Ad^2    I+Ad^2
         (ft)     (ft)    (ft^4)       (ft^2)      (ft)       (ft^4)   (ft^4)
---------------------------------------------------------------------------
NS-1     1.00    40.00      5333         40.0      0.00            0     5333
---------------------------------------------------------------------------
Sum                                                                      5333

Deflection        :   0.084 in    Height            :    12.0 ft
Total Deflection  :   0.084 in

                          Level Height: 24.0 ft
---------------------------------------------------------------------------
                         Same As Previous Level

NS Centroid       :   20.00 ft    EW Centroid       :    0.00 ft
Av                :   40.00 sqft  Moment of Inertia:    5333 ft^4
Deflection        :   0.084 in    Height            :    12.0 ft
Total Deflection  :   0.168 in

---------------------------------------------------------------------------
                                   NS-2
---------------------------------------------------------------------------
                               Same As NS-1

---------------------------------------------------------------------------
                                   NS-3
---------------------------------------------------------------------------

                          Level Height: 12.0 ft
---------------------------------------------------------------------------
                              Centroidal Axis
                                           NS    NS Moment     EW   EW Moment
Name      t        l          Area        Arm       Area       Arm      Area
         (ft)     (ft)       (ft^2)      (ft)     (ft^3)      (ft)    (ft^3)
---------------------------------------------------------------------------
```

NS-3	1.00	20.00	20.0	10.00	200	0.00	0
Sum			20.0		200		0

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : 10.00 ft EW Centroid : 0.00 ft
Av : 20.00 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
NS-3	1.00	20.00	667	20.0	0.00	0	667
Sum							667

Deflection : 0.222 in Height : 12.0 ft
Total Deflection : 0.222 in

Level Height: 24.0 ft

Same As Previous Level

NS Centroid : 10.00 ft EW Centroid : 0.00 ft
Av : 20.00 sqft Moment of Inertia: 667 ft^4
Deflection : 0.222 in Height : 12.0 ft
Total Deflection : 0.444 in

Center of Rigidity

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
NS-1	12.0	5333	40	0.084	11.905	42.05%	0.0	0.000
NS-2	12.0	5333	40	0.084	11.905	42.05%	40.0	476.190
NS-3	12.0	667	20	0.222	4.505	15.91%	80.0	360.360
Sum					28.314			836.551

Centroid from lower left = sum(R*x)/sum(R) : 29.55 ft
Maximum dimension : 80.00 ft
Eccentricity (e) = centroid-(max dimension)/2 : 10.45 ft
e min = 0.05*max. dimension : 4.00 ft
Eccentricity (e) used for torsional analysis : 10.45 ft
e min considered only for seismic analysis.

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
NS-1	24.0	5333	40	0.168	5.952	42.05%	0.0	0.000
NS-2	24.0	5333	40	0.168	5.952	42.05%	40.0	238.095
NS-3	24.0	667	20	0.444	2.252	15.91%	80.0	180.180
Sum					14.157			418.275

Centroid from lower left = sum(R*x)/sum(R) : 29.55 ft
Maximum dimension : 80.00 ft
Eccentricity (e) = centroid-(max dimension)/2 : 10.45 ft
e min = 0.05*max. dimension : 4.00 ft
Eccentricity (e) used for torsional analysis : 10.45 ft
e min considered only for seismic analysis.

Assumptions used:
Em = 144,000 ksf
Ev = 0.4*Em = 57,600 ksf
All wall thicknesses are equal.
Deflections calculated by applying a 1,000 kip load.
Interstory shear wall deflection is calculated based on cantilever action. Deflection at a level is obtained by summing each story's cantilever deflection from grade.
Deflection = P*(h^3)/(3*Em*I)+(1.2*P*h)/(A*Ev)
h = floor to floor height

Name	h (ft)	Rigidity	dx (ft)	R*dx	R*dx*dx	R*dx/ sum(R*dx*dx)
NS-1	12.0	11.905	29.5	351.732	10392.070	0.01519
NS-2	12.0	11.905	10.5	124.459	1301.161	0.00537
NS-3	12.0	4.505	50.5	227.273	11466.942	0.00981
Sum					23160.173	

Name	h (ft)	Rigidity	dx (ft)	R*dx	R*dx*dx	R*dx/ sum(R*dx*dx)
NS-1	24.0	5.952	29.5	175.866	5196.035	0.01519
NS-2	24.0	5.952	10.5	62.229	650.580	0.00537
NS-3	24.0	2.252	50.5	113.636	5733.471	0.00981
Sum					11580.087	

```
Shear distribution      : Fv = V*R/sum(R)
Torsional moment        : Mt = V*e
Torsional component     : Ft = Mt*R*dx/sum(R*dx*dx)
Total shear to element: Ftotal = Fv + Ft
```

F. Save the model as: LAT4.BLD.

SEISMIC FORCES

This chapter is intended to present the procedures, assumptions and limitations incorporated into CASM for the generation of seismic lateral forces based on the equivalent static force methodology presented in the TM 5-809-10 Technical Manual Seismic Design for Buildings, July, 1992. This methodology is an extraction from The Structural Engineer's Association of California (SEAOC) Recommended Lateral Force Requirements and Commentary, 1990 Edition. Symbols, seismic vocabulary and equations used by CASM and illustrated in this tutorial are taken from these documents. The user of CASM is urged to familiarize himself with these documents before attempting to use CASM for seismic force generation.Both of these documents provide guidance to the engineer when special structures or design conditions require the application of a dynamic lateral force procedure. Dynamic analysis is beyond the capabilities of CASM.

Every building's structural system must meet specific requirements outlined in SEAOC regarding configuration, vertical and plan irregularities, combinations of systems, and height limits. Dialog windows can be accessed to help the user test the suitability of a building system to utilize the equivalent static force procedures. Generally, if approximate plan and elevation symmetry exists, not only of the building itself but also its lateral resistance elements, arrangement of openings, and distribution of mass, it will most likely qualify for the static lateral force procedure. Specific SEAOC limitations on height, building period and soil profile within certain seismic zones may require a dynamic analysis procedure, even if all other requirements are met. The engineer should carefully assess all of these issues prior to using CASM.

The information required by CASM to determine equivalent static lateral forces is outlined as follows:

A. Establish Criteria

1. Project dialog window

   a. Lateral load resistance system N/S and E/W: RW

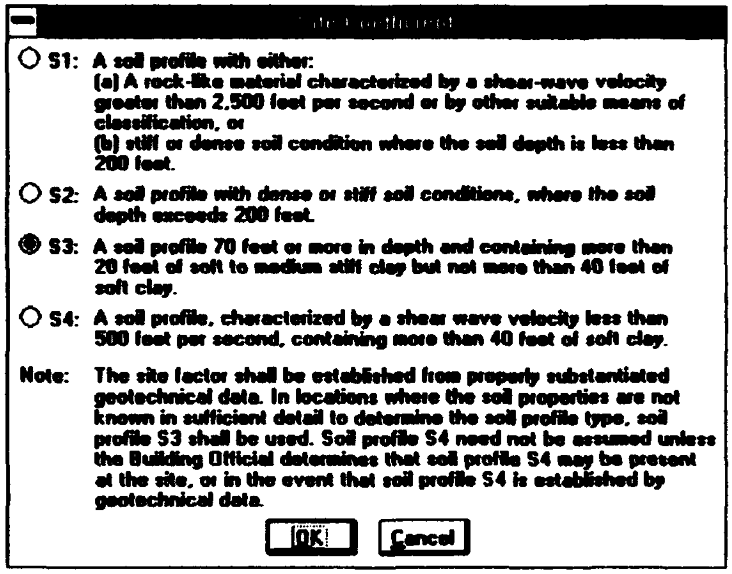
Structural System #1	Rw #5	H #2
A. Bearing Wall System		
1. Light Framed Walls With Shear Panels		
a. Plywood Walls for Structures 3-stories or Less	8	65
b. All Other Light Framed Walls	6	65
2. Shear Walls		
a. Concrete	6	160
b. Masonry	6	160
3. Light Steel Framed Bearing Walls With Tension-Only Bracing	4	65
4. Braced Frames Where Bracing Carries Gravity Loads		
a. Steel	6	160
b. Concrete #3	4	---
c. Heavy Timber	4	65

Notes:

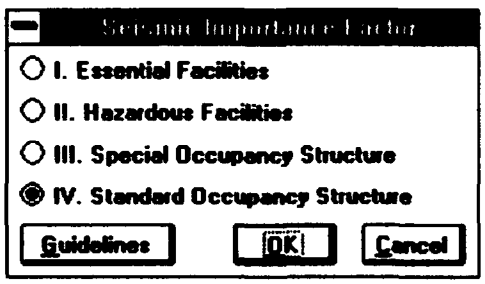
#1 Basic Structural Systems are defined in Section 1.D.6.
#2 H = Height Limit applicable to Seismic Zones 3 and 4. See Section 1.D.7 for exceptions.
#3 Prohibited in Seismic Zones 3 and 4.

OK Cancel

2. **Regional dialog window**
   a. **Geographic zone (map) : Z**
3. **Site dialog window**
   a. **Site geology : S**



   b. **Occupancy : I**



B. Draw the complete building model

C. Draw the complete building structural system

1. **Horizontal planes**
   a. **Draw beam, girder, column, wall and surface elements**
   b. **Diaphragm types**
      (1) **Select flexible or rigid**
2. **Vertical planes**
   a. **Draw braced (trussed or shear walls) or unbraced frames at each desired lateral resistance plane in the north/south and east/west directions.**

D. Assign loads

1. **Dead loads**
   a. **Area loads for floors and roof**

- b. Self-weights of walls and parapets
2. Live loads
   - a. Snow, when the ground snow load exceeds 30 psf
   - b. Occupancy, when storage or warehouse
3. Load combination

E. Generate seismic loads
1. Review spectral plots
2. Review choices made above

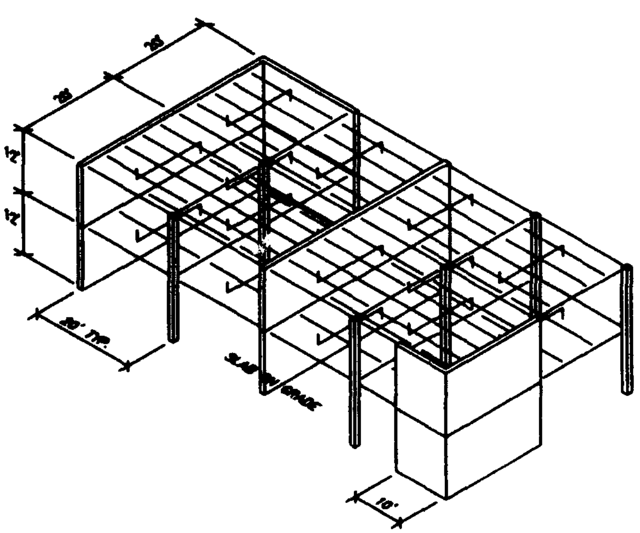
F. Select Lateral Resistance from the Design pull-down menu
1. Select the desired vertical structural plane

G. Review and print analysis output: N/S and E/W directions
1. Total building weight, total base shear
2. Distribution of base shear by floor level, overturning moments by floor level
3. Center of mass by floor level
4. Center of rigidity by level, eccentricity, torsion
5. Shear and overturning moment distribution to resisting elements by floor levels

■ Seismic Design

[illegible]

Given: A two story steel framed administrative building located in Savannah, Georgia. Three equally spaced vertical lateral resistance planes in the north-south direction and two in the east-west direction will be considered. The floor diaphragm will be considered rigid and the roof diaphragm will be considered flexible. This example will utilize masonry shear walls for lateral resistance.



Dead Loads:

Floor: 1-1/2" + 2-1/2" NLWT Concrete Deck
Suspended Ceiling
1" Carpet & Pad
Mechanical - 3.0 psf
Electrical - 1.0 psf
Steel Beams - 3.3 psf

Roof: 1 1/2" Metal Deck - 20 Gage
Suspended 1/2" Drywall Ceiling
5 Ply Built-up Roofing
4" Rigid Insulation
Mechanical - 3.0 psf
Electrical - 1.0 psf
Steel Bar Joists 4' o.c.

Walls: 8" CMU - Medium Weight - Solid Grouted - 78.0 psf

Required: Calculate seismic forces and perform a shear wall lateral resistance analysis in the N-S direction only. Use the dead + seismic load case.

Solution:

A. Establish Criteria.

1. Open the given model SEISMIC1.BLD or enter the following criteria:

Project:	Project Name	: SEISMIC EXAMPLE 1
	City/Installation	: SAVANNAH
	State	GA
	Seismic Load	: TM 5-809-10 1992
	Lateral Resistance System:	
	N-S	: A.2.b.
	E-W	: A.2.b.
Regional:	Seismic Zone	: 2A
Site:	Seismic Importance	: IV
	Seismic Soil Factor	: S3

B. Draw Volumetric Model.

1. The 3D model is drawn in file SEISMIC1.BLD or draw model from the given information.

» **Note: It is necessary to draw the <u>structural</u> walls to be eventually considered as shear walls for lateral resistance under 'Draw Structure'. This should not be confused with planes drawn under 'Draw Model' as walls.**

» **Note: It should be understood that the structural model must include all the lateral resistance elements in the north-south and east-west directions.**

» **Note: Lateral loads will be distributed to the vertical resisting planes according to the vertical resisting element stiffness when a rigid diaphragm is selected. Lateral loads will be distributed to the vertical resisting planes according to tributary width or based on a continuous beam model at the user's choice when a flexible diaphragm is selected.**

C. Develop Independent Load Cases.

1. The loads are already applied in file SEISMIC1.BLD or apply the loads given above.

» **Note: All the floor and roof dead loads must be assigned before seismic forces can be calculated.**

» **Note: Self-weight of beams and columns can be included in the building dead weight by either of the following methods: 1. smeared into the area loads or 2. entered separately in the self-weight dialog windows as they appear in steps D6 and D7.**

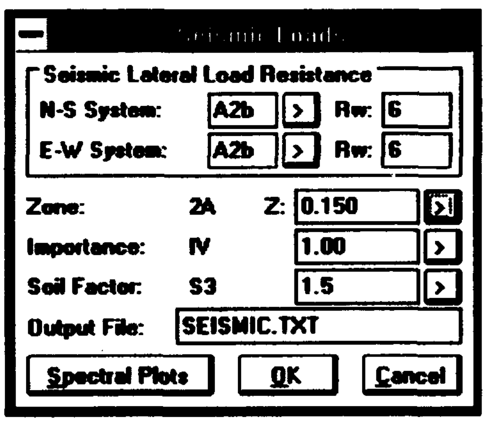
» **Note:Wall and parapet dead loads are assigned as linear wall loads.**

D. Calculate total seismic forces at each level.

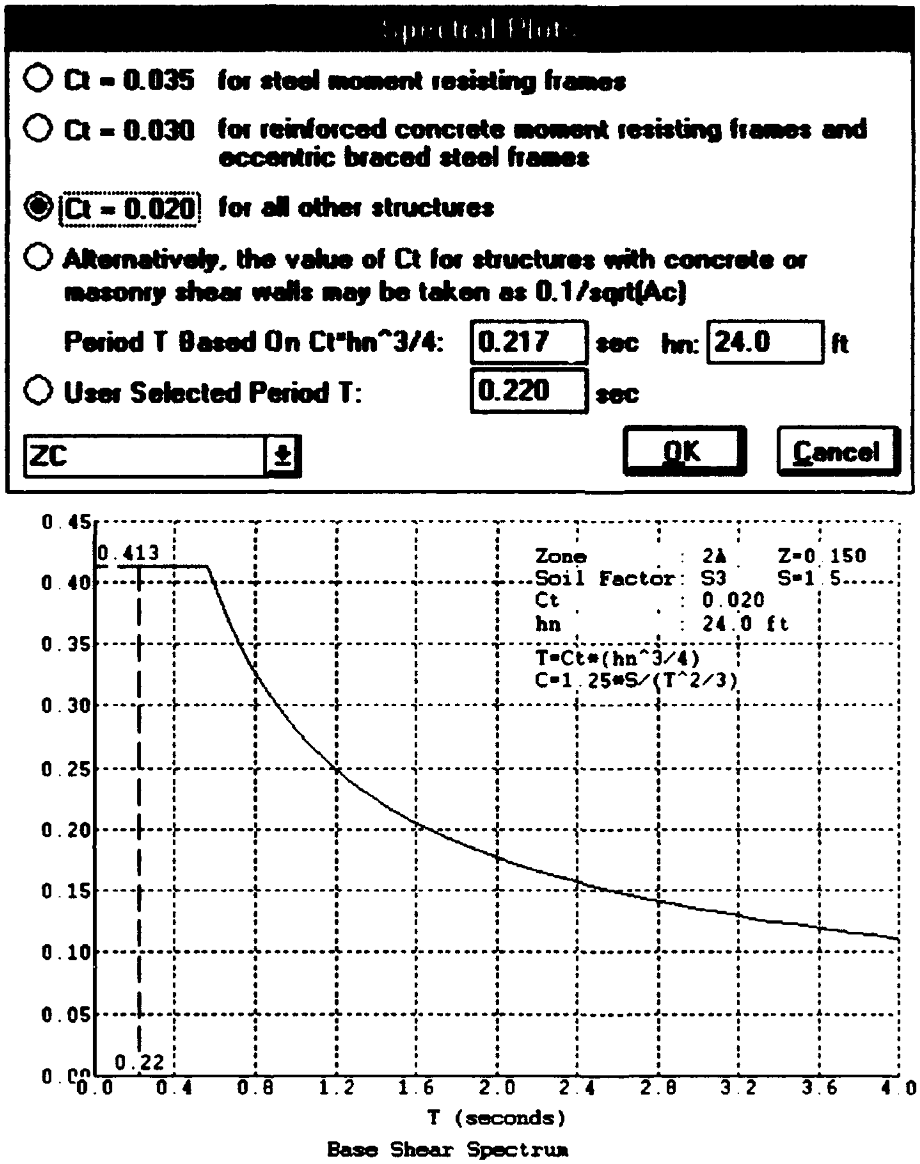
1. Select the LOADS AND DESIGN tool palette.
2. Select the Load Combination: DEAD + SEISMIC.

» **Note:In order to generate seismic loads, a load combination must be selected. This is required since seismic forces may include a percentage of occupancy live load and roof snow load under certain conditions.**

3. Select SEISMIC from the Loads pull-down menu.



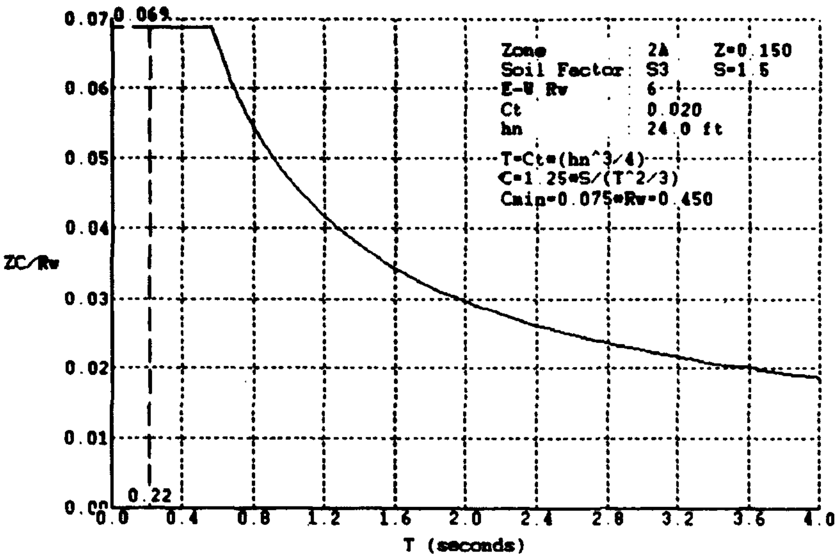
a. Click on SPECTRAL PLOTS to review the Base Shear Spectrum and the Design Base Shear Coefficient Spectrum.



(1) Select Ct to have building period (T) calculated by equation in code or input known period (T). Select CT = 0.020 FOR ALL OTHER BUILDINGS for this example.

(2) Select ZC/RW - EW to view the Design Base Shear Coefficient Spectrum.

» **Note: If the Rw is different in both the north-south and east-west directions, separate ZC/Rw directions will appear. Select the direction under consideration to view the correct plot.**



Design Base Shear Coefficient Spectrum

» **Note: Both spectrums can be printed using the Print Screen command.**

(3) Select OK when you are finished viewing the spectrums.

b. Review seismic criteria previously selected and make any final corrections.

c. Select OK to begin calculating total seismic forces at each level.

4. Review the dialog windows describing specific criteria regarding configuation, as well as vertical and plan irregularities.

Plan Structural Irregularities

Irregularity Type and Definition	Reference Section
A. Torsional Irregularity, to be considered when diaphragms are not flexible Torsional irregularity shall be considered to exist when the maximum story drift, computed including accidental torsion, at one end of the structure transverse to an axis is more than 1.2 times the average of the story drifts of the two ends of the structure.	1.E.6d, 1.H.1c, 1.H.2i(4)
B. Reentrant Corners Plan configurations of a structure and its lateral force-resisting system contain reentrant corners, where both projections of the structure beyond a reentrant corner are greater than 15 percent of the plan dimension of the structure in the given direction.	1.H.2i(4), 1.H.2i(5)
C. Diaphragm Discontinuity Diaphragms with abrupt discontinuities or variations in stiffness, including those having cutout or open areas greater than 50 percent of the gross enclosed area of the diaphragm.	1.H.2i(4)
D. Out-of-Plane Offsets Discontinuities in a lateral force path, such as out-of-plane offsets of the vertical elements.	1.E.7b, 1.H.2i(4)
E. Nonparallel Systems The vertical lateral load resisting elements are not parallel to nor symmetric about the major orthogonal axes of the lateral force resisting system.	1.H.1c

OK Cancel

Irregularity Type and Definition	Reference Section
A. Stiffness Irregularity - Soft Story A soft story is one in which the lateral stiffness is less than 70 percent of that in the story above or less than 40 percent of the combined stiffness of the three stories above.	1.D.8b(2)
B. Weight (mass) Irregularity Mass irregularity shall be considered to exist where the effective mass of any story is more than 150 percent of the effective mass of an adjacent story. A roof which is lighter than the floor below need not be considered a mass irregularity.	1.D.8b(2)
C. Vertical Geometric Irregularity Vertical geometric irregularity shall be considered to exist where the horizontal dimension of the lateral force resisting system in any story is more than 130 percent of that in an adjacent story. One-story penthouses need not be considered.	1.D.8b(2)
D. In-Plane Discontinuity in Vertical Lateral Force Resisting Element An in-plane offset of the lateral load resisting elements greater than the length of those elements.	1.E.7b
E. Discontinuity in Capacity - Weak Story A weak story is one in which the story strength is less than 80 percent of that in the story above. The story strength is the total strength of all seismic resisting elements sharing the story shear for the direction under consideration.	1.D.9a

[OK] [Cancel]

a. Select Cancel if any described irregularity exists. Seismic forces will not be calculated. A dynamic analysis, which is beyond the capabilities of CASM, will be required.

b. Select OK if no irregularities exist and to continue the equivalent static force procedure. The Ct dialog window will appear.

5. Select CT = 0.020 FOR ALL OTHER BUILDINGS to have the period calculated from the code prescribed equation and select OK.

>> **Note: The user has the option of entering a known period (T) to override calculation by the code equation.**

○ Ct = 0.035 for steel moment resisting frames

○ Ct = 0.030 for reinforced concrete moment resisting frames and eccentric braced frames

◉ Ct = 0.020 for all other buildings

○ Alternatively, the value of Ct for structures with concrete or masonry shear walls may be taken as 0.1/sqrt(Ac)

Period T Based On Ct*hn^3/4: 0.22 sec

○ User Selected Period T: 0.22 sec

[OK] [Cancel]

6. Enter an Estimated Beam Self Weight of 20.0 plf and select OK.

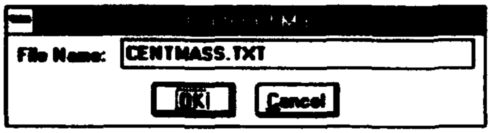
>> **Note: This is based on the fact that the girder self weights have not been smeared into area loads.**

» **Note: The lengths of all girders on the grid lines are multiplied by the self-weight to be included in the mass calculations. The weights of the beams and joists off the grid lines are already smeared into the floor and roof dead loads.**

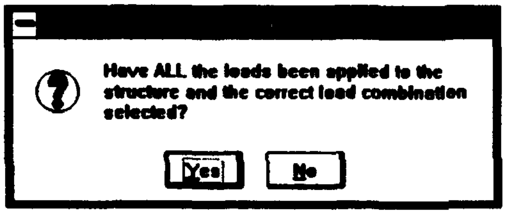
7. Enter an Estimated Column Self Weight of 24.0 plf and select OK.

» **Note: This is based on the fact that the column weights have not been smeared into the area loads.**

8. Enter an appropriate Center Of Mass File Name in the Center Of Mass dialog window and select OK.



9. A Seismic dialog window will prompt you to verify that all loads have been applied and the correct load combination selected. Select YES to continue. Total seismic forces at each level are now calculated.
10. Review the seismic force at each level and the center of mass output files.



a. Select PRINT DATA from the File pull-down menu. The Print Data dialog window will appear.
b. Select only the SEISMIC and CENTER OF MASS output files at this time.
c. Select either PRINT TO PRINTER or PRINT TO FILE and select OK.
d. Review total building weight, total base shear and total force and overturning moments at each level. Review the center of mass at each level in both the north-south and east-west directions. When you are finished reviewing the output, return back to CASM.

Example 1 Sample output:

```
Project      : Seismic Example 1
Location     : Savannah
Seismic Code: TM 5-809-10 1991
Time         : Wed Oct 09, 1991   3:57 PM

**************************** Seismic Analysis ******************************

 3. Roof                                           :      135.1 k
 2. Second Floor                                   :      277.8 k
-----------------------------------------------------------------------
Total Building Weight (W)                          :      412.9 k

***************************** N - S and E - W ******************************

Zone: 2A: Z = 0.150
Importance Category: IV: I = 1.00
Soil Factor: S3: S = 1.5
System: A2b: Rw = 6
Ct = 0.020
hn = 24.0 ft
```

T = Ct*hn^3/4 = 0.22 sec
C = 1.25*S/T^2/3 = 5.15 > 2.75
C = 2.75
C/Rw = 0.458 > 0.075
W = 412.9 k
V = Z*I*C*W/Rw

V = 28.4 k

T < 0.7 sec

Ft = 0.0 k

V-Ft = 28.4 k

Level	h (ft)	Floor to Floor h (ft)	w (k)	sum(w) (k)	w*h (kft)	w*h/ sum(w*h)	F (k)	sum(F) V (k)
						Ft =	0.0	
3	24.0		135		3242	0.493	14.0	
		12.0		135				14.0
2	12.0		278		3333	0.507	14.4	
		12.0		413				28.4
1	0.0							
Sum			413		6576	1.000	28.4	

Level	h (ft)	Floor to Floor h (ft)	w (k)	sum(w) (k)	sum(F) V (k)	OTM (kft)	sum(OTM) (kft)	Ft+sum(F)/ sum(w)
3	24.0		135					
		12.0		135	14.0	168		0.104
2	12.0		278				168	
		12.0		413	28.4	341		0.069
1	0.0						509	
Sum			413			509		

Project : Seismic Example 1
Location : Savannah
Time : Wed Oct 09, 1991 3:57 PM

****************************** Center Of Mass ******************************

Roof -- 24.00 ft

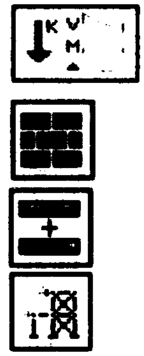
Name	Weight (k)	NS (ft)	NS*Weight (kft)	EW (ft)	EW*Weight (kft)
Wall Type 1	18.7	20.0	374.4	0.0	0.0
Wall Type 1	9.4	40.0	374.4	10.0	93.6
Wall Type 1	18.7	20.0	374.4	40.0	748.8
Wall Type 1	9.4	10.0	93.6	80.0	748.8
Wall Type 1	4.7	0.0	0.0	75.0	351.0
Roof Type 1	67.2	20.0	1344.0	40.0	2688.0
Beam Self Weight	6.2	20.0	124.0	40.0	248.0
Column Self Weight	0.9	20.0	17.3	40.0	34.6
Sum	135.1		2702.1		4912.8

N-S Center Of Mass: 20.00 ft
E-W Center Of Mass: 36.36 ft

Second Floor -- 12.00 ft

Name	Weight (k)	NS (ft)	NS*Weight (kft)	EW (ft)	EW*Weight (kft)
Wall Type 1	37.4	20.0	748.8	0.0	0.0
Wall Type 1	18.7	40.0	748.8	10.0	187.2
Wall Type 1	37.4	20.0	748.8	40.0	1497.6
Wall Type 1	18.7	10.0	187.2	80.0	1497.6
Wall Type 1	9.4	0.0	0.0	75.0	702.0
Floor Type 1	148.2	20.0	2963.2	40.0	5926.4
Beam Self Weight	6.2	20.0	124.0	40.0	248.0
Column Self Weight	1.7	20.0	34.6	40.0	69.1
Sum	277.8		5555.4		10127.9

N-S Center Of Mass: 20.00 ft
E-W Center Of Mass: 36.46 ft



E. Establish element parameters to perform the lateral analysis.

1. Select the LOADS AND DESIGN tool palette.
2. Select material: MASONRY.

F. Preliminary Lateral Analysis

1. Select the load combination: DEAD + SEISMIC.
2. Select LATERAL RESISTANCE from the Design pull-down menu.
3. Select the shear wall location NS-2 on grid line C. The 2D view of the vertical resistance plane will appear.
4. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + E.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL=DECK+SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
5. Select CONTINUOUS BEAM MODEL from the Flexible Diaphragm dialog window to distribute the seismic forces on the flexible diaphragm roof plane using the continuous beam model. Select OK and the Rigid Horizontal Diaphragm Calculations dialog window appears.
6. Turn ON CONSIDER PERPENDICULAR WALL ELEMENTS AS FLANGES TO THE LATERAL RESISTANCE SYSTEM, turn OFF INCLUDE OPENINGS IN SHEAR WALL STIFFNESS, enter an appropriate File Name and select OK. The load calculations will begin.

» **Note: The option exists when floor or roof diaphragms are considered rigid to either 1) assume the perpendicular wall elements are not attached for the computation of wall cross sectional properties, or 2) include perpendicular wall elements as attached for the computation of wall cross sectional properties. Only 6 times the wall thickness will be used for the attached perpendicular length in the calculations. This permits treating walls as L, C, T, or Box (back to back C's) shaped.**

» **Note: Each lateral resistance plane is analyzed for a one thousand kip lateral force to compare stiffness for seismic load distribution on the rigid diaphragm floor plane.**

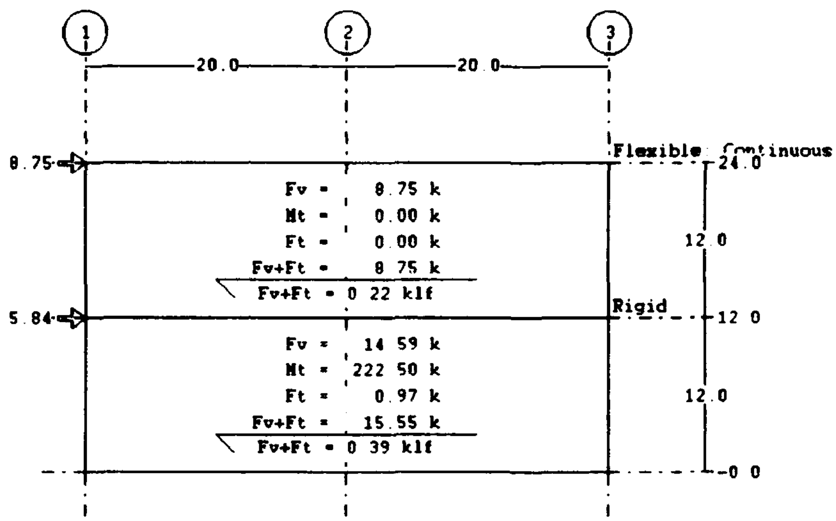
» **Note: A one thousand kip force is used to compare shear wall stiffnesses. A one kip force is used to compare rigid frame stiffnesses. A one kip force is used to compare trussed bracing stiffnesses.**

7. Review the loads on the shear wall by scrolling through the list in the View Loads dialog window. Select OK when you are finished viewing the load diagrams. The Seismic Lateral Resistance Locations dialog window will appear.

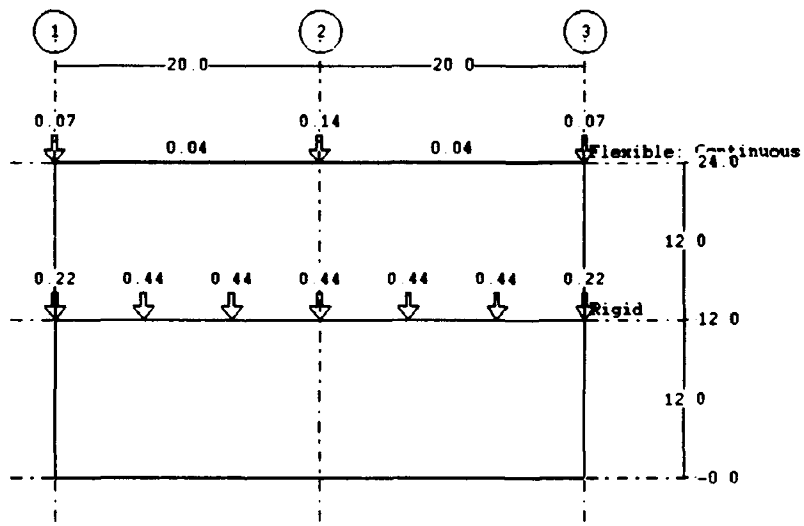
» **Note: The two story shear wall elevation illustrates the applied seismic forces at each level as a concentrated load. The shear (Fv) at each level plus the shear associated with any torsion (Ft) is tabulated and divided by the length of the shear wall to calculate the shear load per lineal foot carried by the wall. The heading includes the percentage of the total seismic force distibuted to this shear wall at the rigid floor diaphragm level.**

» **Note: The self weight of the shear wall is not included in the dead load diagram.**

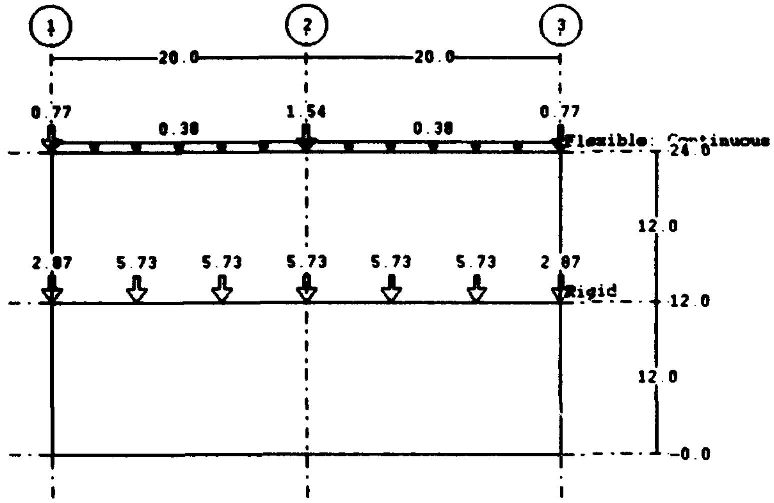
» **Note: This version of CASM does not perform an analysis of the shear wall.**



1.00 Seismic (klf) -- NS-2 -- F, 41%

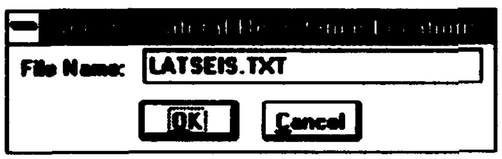


1.00 Dead (klf)



1.00 Superimposed Dead (klf)

8. **Enter an appropriate File Name and select OK. The shear and overturning moments distributed to each resisting element at each level are calculated.**



9. **Review the Rigid Horizontal Diaphragm Calculations and the Seismic Lateral Resistance Locations calculations.**
   a. **Select PRINT DATA from the File pull-down menu.**
   b. **Select only the RIGID DIAPHRAGM and SEISMIC RESISTANCE output files.**
   c. **Select either PRINT TO PRINTER or PRINT TO FILE and select OK.**
   d. **Review the cross sectional properties for each resisting element taken about the north-south and east-west centroidal axes.**

» **Note: The stiffness of resisting elements are computed at each level.**

» **Note: The cross sectional properties of resisting elements that might occur repetitively in other examples are not duplicated in the output file.**

» **Note: For this example, there are three resisting elements in the north-south direction and two in the east-west direction.**

   e. **Review the center of rigidity calculations in the north-south and east-west directions. When you are finished reviewing the output, return back to CASM.**

» **Note: The assumptions used in the calculation of interstory deflection are included in the output.**

» **Note: The equations for torsional moment and distribution of torsional shear are included in the output with the magnitudes shown on the wall elevations previously viewed.**

Example 1 Sample Output:

Project : Seismic Example 1
Location : Savannah
Time : Fri Sep 27, 1991 12:53 PM

***************** Rigid Horizontal Diaphragm Calculations *****************

NS-1

Level Height: 12.0 ft

Centroidal Axis

Name	t (ft)	l (ft)	Area (ft^2)	NS Arm (ft)	NS Moment Area (ft^3)	EW Arm (ft)	EW Moment Area (ft^3)
NS-1	0.67	40.33	26.9	20.17	542	0.00	0
EW-1	0.67	4.00	2.7	40.00	107	2.33	6
Sum			29.6		649		6

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : 21.96 ft EW Centroid : 0.21 ft
Av : 26.89 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
NS-1	0.67	40.33	3645	26.9	-1.79	86	3731
EW-1	4.00	0.67	0	2.7	18.04	868	868
Sum							4600

Deflection : 0.122 in Height : 12.0 ft
Total Deflection : 0.122 in

NS-2

Level Height: 12.0 ft

Centroidal Axis

Name	t (ft)	l (ft)	Area (ft^2)	NS Arm (ft)	NS Moment Area (ft^3)	EW Arm (ft)	EW Moment Area (ft^3)
NS-2	0.67	40.00	26.7	20.00	533	0.00	0
Sum			26.7		533		0

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : 20.00 ft EW Centroid : 0.00 ft
Av : 26.67 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
NS-2	0.67	40.00	3556	26.7	0.00	0	3556
Sum							3556

Deflection : 0.126 in Height : 12.0 ft
Total Deflection : 0.126 in

NS-3

Level Height: 12.0 ft

Centroidal Axis

Name	t (ft)	l (ft)	Area (ft^2)	NS Arm (ft)	NS Moment Area (ft^3)	EW Arm (ft)	EW Moment Area (ft^3)
NS-3	0.67	20.33	13.6	10.17	138	0.00	0
EW-2	0.67	4.00	2.7	0.33	1	-2.33	-6
Sum			16.2		139		-6

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : 8.55 ft EW Centroid : -0.38 ft
Av : 13.56 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
NS-3	0.67	20.33	467	13.6	1.62	35	502
EW-2	4.00	0.67	0	2.7	-8.22	180	180
Sum							683

Deflection : 0.292 in Height : 12.0 ft
Total Deflection : 0.292 in

EW-1

Level Height: 12.0 ft

Centroidal Axis

Name	t (ft)	l (ft)	Area (ft^2)	NS Arm (ft)	NS Moment Area (ft^3)	EW Arm (ft)	EW Moment Area (ft^3)
EW-1	0.67	20.33	13.6	0.00	0	10.17	138
NS-1	0.67	4.00	2.7	-2.33	-6	0.33	1
Sum			16.2		-6		139

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : -0.38 ft EW Centroid : 8.55 ft
Av : 13.56 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
EW-1	0.67	20.33	467	13.6	1.62	35	502
NS-1	4.00	0.67	0	2.7	-8.22	180	180
Sum							683

Deflection : 0.292 in Height : 12.0 ft
Total Deflection : 0.292 in

EW-2

Level Height: 12.0 ft

Centroidal Axis

Name	t (ft)	l (ft)	Area (ft^2)	NS Arm (ft)	NS Moment Area (ft^3)	EW Arm (ft)	EW Moment Area (ft^3)
EW-2	0.67	10.33	6.9	0.00	0	5.17	36
NS-3	0.67	4.00	2.7	2.33	6	10.00	27
Sum			9.6		6		62

Centroid = sum(MomentArea)/sum(Area)
NS Centroid : 0.65 ft EW Centroid : 6.52 ft
Av : 6.89 sqft

Moment of Inertia

Name	b (ft)	h (ft)	bh^3/12 (ft^4)	Area (ft^2)	d (ft)	Ad^2 (ft^4)	I+Ad^2 (ft^4)
EW-2	0.67	10.33	61	6.9	-1.35	13	74
NS-3	4.00	0.67	0	2.7	3.48	32	32
Sum							106

Deflection : 0.887 in Height : 12.0 ft
Total Deflection : 0.887 in

Center of Rigidity

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
NS-1	12.0	4600	27	0.122	8.196	41.90%	0.2	1.726
NS-2	12.0	3556	27	0.126	7.937	40.57%	40.0	317.460
NS-3	12.0	683	14	0.292	3.429	17.53%	79.6	273.004
Sum					19.562			592.190

Centroid from lower left = sum(R*x)/sum(R) : 30.27 ft
Center of mass from lower left : 36.46 ft
Eccentricity (e) : 6.19 ft
Maximum dimension : 80.00 ft
e min = 0.05*max. dimension : 4.00 ft
Eccentricity (e) used for torsional analysis : 6.19 ft

Name	h (ft)	I (ft^4)	Av (ft^2)	Deflection (in)	Rigidity	R/ sum(R)	x (ft)	R*x
EW-1	12.0	683	14	0.292	3.429	75.26%	39.6	135.845
EW-2	12.0	106	7	0.887	1.127	24.74%	0.7	0.734
Sum					4.556			136.579

Centroid from lower left = sum(R*x)/sum(R) : 29.98 ft
Center of mass from lower left : 20.00 ft
Eccentricity (e) : 9.98 ft
Maximum dimension : 40.00 ft
e min = 0.05*max. dimension : 2.00 ft
Eccentricity (e) used for torsional analysis : 9.98 ft

Assumptions used:
Em = 144,000 ksf
Ev = 0.4*Em = 57,600 ksf
All wall thicknesses are equal.
Deflections calculated by applying a 1,000 kip load.
Interstory shear wall deflection is calculated based on cantilever action. Deflection at a level is obtained by summing each story's

```
    cantilever deflection from grade.
 Deflection = P*(h^3)/(3*Em*I)+(1.2*P*h)/(A*Ev)
    h = floor to floor height
```

Name	h (ft)	Rigidity	dx (ft)	R*dx	R*dx*dx	R*dx/ sum(R*dx*dx)
NS-1	12.0	8.196	30.1	246.400	7407.324	0.01385
NS-2	12.0	7.937	9.7	77.200	750.947	0.00434
NS-3	12.0	3.429	49.3	169.199	8348.919	0.00951
EW-1	12.0	3.429	9.6	33.060	318.737	0.00186
EW-2	12.0	1.127	29.3	33.060	969.446	0.00186
Sum					17795.373	

```
Shear distribution      : Fv = V*R/sum(R)
Torsional moment        : Mt = V*e
Torsional component     : Ft = Mt*R*dx/sum(R*dx*dx)
Total shear to element: Ftotal = Fv + Ft
```

```
Project      : Seismic Example 1
Location     : Savannah
Seismic Code: TM 5-809-10 1991
Time         : Wed Oct 09, 1991  4:02 PM
```

****************** Seismic Lateral Resistance Locations *******************

NS-1 -- F, 42%

Level	h (ft)	Floor to Floor h (ft)	F (k)	sum(F) V (k)	OTM (kft)	sum(OTM) (kft)
3	24.0		14.0			
		12.0		14.0	168	
2	12.0		14.4			168
		12.0		28.4	341	
1	0.0					509
Sum			28.4		509	

NS-2 -- F, 41%

Level	h (ft)	Floor to Floor h (ft)	F (k)	sum(F) V (k)	OTM (kft)	sum(OTM) (kft)
3	24.0		14.0			
		12.0		14.0	168	
2	12.0		14.4			168
		12.0		28.4	341	
1	0.0					509
Sum			28.4		509	

NS-3 -- F, 18%

Level	h (ft)	Floor to Floor h (ft)	F (k)	sum(F) V (k)	OTM (kft)	sum(OTM) (kft)
3	24.0		14.0			
		12.0		14.0	168	
2	12.0		14.4			168
		12.0		28.4	341	
1	0.0					509
Sum			28.4		509	

G. Save the model as: SEISMIC1.BLD.

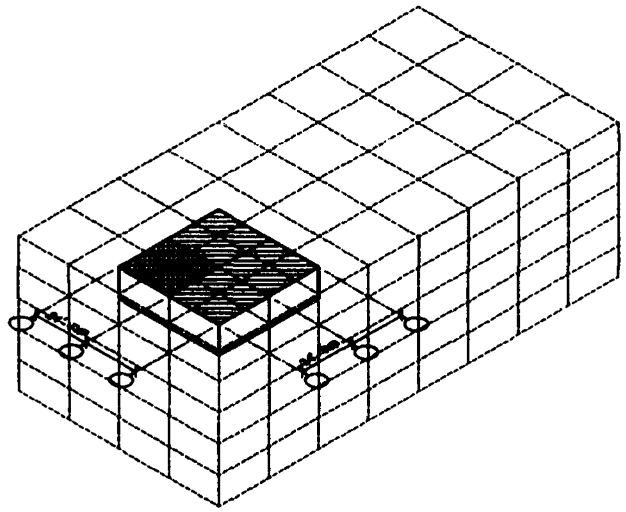
QUANTITY TAKEOFFS

This chapter is intended to describe the procedures incorporated in CASM to accumulate quantities appropriate for preliminary cost estimating. Information stored from the geometric model and from the selection of members utilizing the CASM Spreadsheets is used for automatic generation of material quantities. When spreadsheets do not exist to design a particular type of element, the user must manually enter the necessary information. Three levels of quantity take-offs might be considered useful by the engineer during his preliminary comparison of structural systems: (1) one typical interior bay, (2) one typical level, or (3) the entire building's structural system. The choice is linked to the number of different members the user cares to design, as well as time and storage considerations.

TYPICAL INTERIOR BAY - SYSTEM COMPARISON

Example 1:

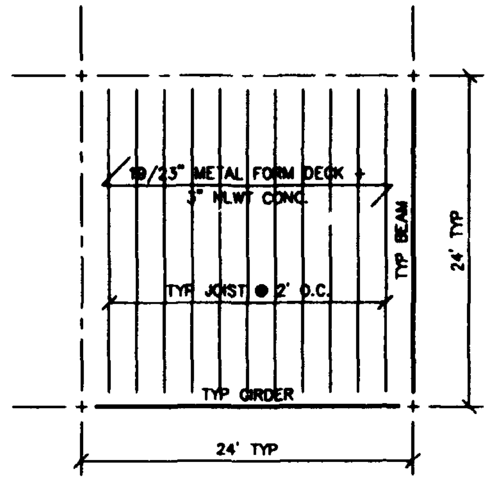
Given: A five story, 4 bay by 8 bay bank and office building located in Champaign, Illinois with dimensions as illustrated below. Occupancy live load shall be 50 psf.



The following three floor framing schemes will be considered for this comparison:

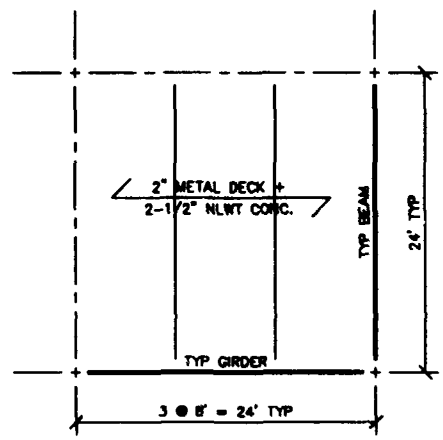
A. Steel Bar Joists 2' o.c.

Dead Loads:
- 3" NLWT Concrete
- 19/32" Metal Form Deck
- Suspend Accoustical Tile Ceiling
- Mechanical - 3 psf.
- Electrical - 2 psf.
- Partitions - 10 psf.
- Carpet and Pad



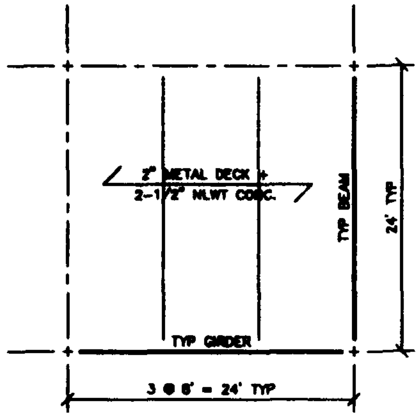
B. Steel Non-Composite Beams at 8' o.c.

Dead Loads: 2-1/2" NLWT Concrete
2" Composite Deck - 20 ga.
Suspend Accoustical Tile Ceiling
Mechanical - 3 psf.
Electrical - 2 psf.
Partitions - 10 psf.
Carpet and Pad



C. Steel Composite Beams/Slab at 8' o.c.

Dead Loads: 2-1/2" NLWT Concrete
2" Composite Deck - 20 ga.
Suspend Accoustical Tile Ceiling
Mechanical - 3 psf.
Electrical - 2 psf.
Partitions - 10 psf.
Carpet and Pad



Required: Determine the material quantities for a typical interior bay's floor framing for the three given schemes.

Solution:

SCHEME A Steel Bar Joists

A. Establish Criteria

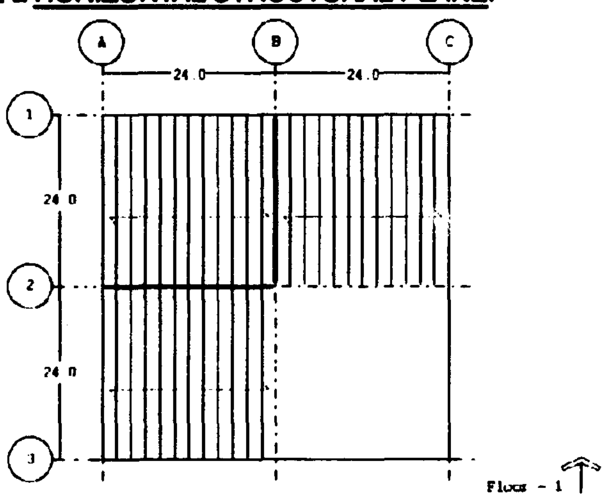
1. **Enter the following Project criteria.**

Project:	**Project Name**	: BANK - SCHEME A
	City/Installation	: CHAMPAIGN
	State	: IL

B. Draw Volumetric Model and Structure

>> Note: It is only necessary to extract a 2 bay by 2 bay by 1 story high model from the building to design a typical interior bay.

1. **Draw a CUBE with the dimensions of 48' by 48' by 13' high.**
2. **DEFINE GRID as 24' by 24'.**
3. **Select a HORIZONTAL STRUCTURAL PLANE.**

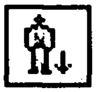




4. Draw the NARROWLY SPACED Linear joist elements 2' o.c. as shown above.
5. Draw the ONE-WAY Surface elements as shown above.
6. Draw the two WIDELY SPACED Linear elements, one a beam, the other a girder, as shown above.

>> **Note: The rolled steel beam and girder quantities for one bay include only one beam and only one girder.**

C. Develop Independent Load Cases




1. Create and assign the OCCUPANCY LIVE LOAD of 50 psf to the entire floor plane.
2. Create and assign the FLOOR DEAD LOAD to the entire floor plane.

Deck	: 3" Metal Form Deck + NLWT Concrete
Ceiling	: Suspend Accoustical Tile Ceiling
Mechanical	: 3 psf.
Electrical	: 2 psf.
Partitions	: 101-200 plf - 10 psf.
Finish	: Carpet and Pad

>> **Note: The joist self weight will be smeared into the area load during the joist design.**

3. Save the model as QUANT1.BLD for use in schemes B and C.

D. Establish element parameters to design a typical open-web steel joist.

1. Select STEEL from the Material pull-down menu.
2. Select OPEN-WEB JOISTS - K from the Surf/Line pull-down menu.
3. Select any handle in the corner bay A1-B2.
4. Review the Linear Elements and the Element Attributes dialog windows.

E. Preliminary analysis of a typical open-web steel joist.


1. Select the Load Combination: DEAD + LIVE.
2. Select PRELIMINARY from Design pull-down menu.
3. Select Units options:
   a. Select units of FEET and POUNDS.
   b. Verify load combination of D + L.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL=DECK + SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
4. Select the connectivity as a HINGE and a ROLLER.
5. Enter an Estimated Self Weight of 7.0 psf and turn ON UPDATE AREA STRUCTURE LOADS.
6. Enter an appropriate Analysis File Name. Preliminary analysis begins.
7. View the shear, moment, deflection, loads and reactions diagrams.

F. **Preliminary design of a typical open-web steel joist.**

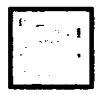
1. Select EXECUTE EXCEL in the Excel Data dialog window. The CASM program will become an icon and Excel will be executed loading the open-web steel joist design spreadsheet.
2. Review the open-web steel joist design options listed.
3. Select the 16K2 since it is the lightest possible joist and send the member size to CASM.

» **Note: The actual weight (5.5 plf) of the bar joist will be used in the quantity take-off calculations. The estimated joist weight (7.0 plf) will still be used for the applied load to the beams, girders and columns. It is possible to revise the joist weight in the floor dead load based on the engineer's judgement. It will not be done in this example since the weights are reasonably close.**

4. Return to CASM. The selected joist size will be displayed on the floor plane.

G. **Analyze and design the beam parallel to the joists on grid line B.**

» **Note: Follow similar procedure as used for the joist design.**



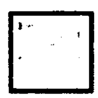
1. Analyze a Widely Spaced Steel Rolled Section with an estimated self weight of 22.0 plf. Do NOT update the area structure loads. Assume a Hinge and a Roller for connectivity.
2. Design the beam in Excel and select a W 12 x 14, sending the member size to CASM.
3. Return to CASM. The selected beam size will be displayed on the floor plane.

H. **Analyze and design the girder perpendicular to the joists on grid line 2.**

» **Note: Follow similar procedure as used for the beam design.**

1. Analyze a Widely Spaced Steel Rolled Section with an estimated self weight of 50.0 plf. Assume a Hinge and a Roller for connectivity.
2. Design the girder in Excel and select a W 21 x 50, sending the member size to CASM.
3. Return to CASM. The selected girder size will be displayed on the floor plane.

I. **Analyze and design the metal form deck.**

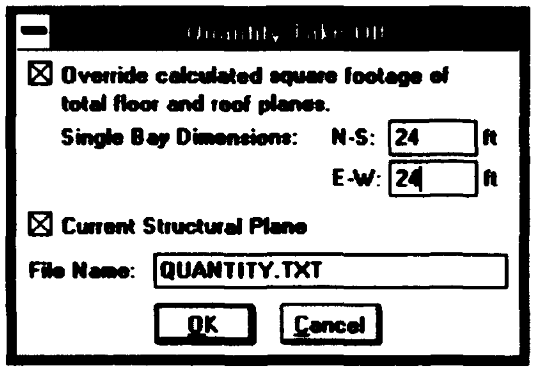


1. Change the Load Combination to D (Dead Load 1.0).
2. Select Form Deck from the Surface/Linear menu.
3. Select Preliminary from the Design pull-down menu.
4. Select units as feet and pounds.
5. Do not include superimposed dead load.
6. Select EXECUTE EXCEL.

7. Select NLWT concrete (145 psf) and 3 inch slab depth from the Member menu.
8. Select formdeck size from spreadsheet as 9/16"-26 GA.
9. Check box to send member size to CASM.
10. Select Return to CASM from the File pull-down menu.

J. **Perform quantity take-off for the typical interior floor bay.**

1. Select QUANTITY TAKE-OFF from the Design pull-down menu. The Quantity Take-off dialog window will appear.



2. Turn ON OVERRIDE CALCULATED SQUARE FOOTAGE OF TOTAL FLOOR AND ROOF PLANES.

» **Note: The CASM default calculates the total square footage based on the size of the model created. It is necessary to override this procedure when a single bay quantity take-off is required. The quantites per square foot will then be correctly calculated for one bay.**

3. Enter the Single Bay Dimensions of 24.0 feet by 24.0 feet.
4. Turn ON CURRENT STRUCTURAL PLANE.
5. Enter an appropriate File Name.
6. Select OK to perform the quantity take-off. Notepad will automatically be executed loading quantity take-off output file.

7. **Review the quantity take-off.**

>> **Note: Structural elements drawn but not designed appear on the output without descriptions or weights.**

Scheme A Sample Output

```
Project       : Bank - Scheme A
Location      : Champaign
Time          : Mon Sep 30, 1991  10:47 AM

**************************** Quantity Take-off ****************************

---------------------------------------------------------------------------
                                 Roof - 2
---------------------------------------------------------------------------
                Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft

---------------------------------------------------------------------------
                                 Floor - 1
---------------------------------------------------------------------------
                Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft

STEEL: Narrowly Spaced Elements
---------------------------------------------------------------------------
                                                     Weight/         Total
Description                        Length  Weight   Element   No.   Weight
                                    (ft)    (plf)    (lbs)           (lbs)
---------------------------------------------------------------------------
16K2                                24.0     5.5     132.0    11     1452
                                    24.0     0.0       0.0    22        0
---------------------------------------------------------------------------
Sum                                                                   1452

Total Weight            :    0.7 tons
Weight Per Square Foot  :    2.5 psf

STEEL: Widely Spaced Elements
---------------------------------------------------------------------------
                                                     Weight/         Total
Description                        Length  Weight   Element   No.   Weight
                                    (ft)    (plf)    (lbs)           (lbs)
---------------------------------------------------------------------------
W 21 x 50                           24.0    50.0    1200.0     1     1200
W 12 x 14                           24.0    14.0     336.0     1      336
---------------------------------------------------------------------------
Sum                                                                   1536

Total Weight            :    0.8 tons
Weight Per Square Foot  :    2.7 psf

STEEL: Surface Elements
---------------------------------------------------------------------------
                           Total                     Conc    Conc    Total  Weight
Description                Depth   Area   Weight   Weight  Weight           Conc
                           (in)   (sqft)   (psf)    (pcf)   (psf)   (lbs)  (lbs)
---------------------------------------------------------------------------
9/16-26 GA FRM DK + 4" NLK  3.0     576     0.9    145.0    32.0     518  18432
                            0.0    1152     0.0      0.0     0.0       0      0
---------------------------------------------------------------------------
Sum                                                                  518  18432

Concrete Cubic Yards    :    4.7
Total Weight            :    0.3 tons
```

K. Save the model as: QUANT1A.BLD.

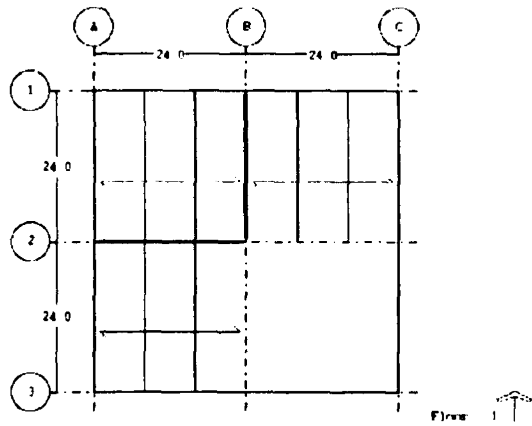
SCHEME B: Steel Non-Composite Beams

A. Establish Criteria

1. OPEN the file QUANT1.BLD.
2. Change the Project Name to BANK - SCHEME B.

B. Draw Structure

1. Select a HORIZONTAL STRUCTURAL PLANE.



2. Delete the NARROWLY SPACED Linear joist elements.
3. Draw the two third point WIDELY SPACED Linear elements as shown above.

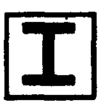
C. Develop Independent Load Cases

1. Modify the following FLOOR DEAD LOAD components.

Deck:	2" Metal Deck + 2-1/2" NLWT Concrete
Structure:	Steel Beams - 0.0 plf

» **Note: The beam self weight will be smeared into the area load during the beam design.**

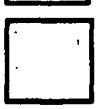
2. Select SAVE in the Floor (DL) dialog window to update the floor area dead load.
3. Save the model as QUANT1.BLD.



D. Establish element parameters to design a typical third point steel beam.

1. Select STEEL from the Material pull-down menu.
2. Select ROLLED SECTIONS from the Surf/Line pull-down menu.
3. Select either handle in the corner bay A1-B2.
4. Review the Linear Elements and the Element Attributes dialog windows.

E. Preliminary analysis of a typical third point steel beam.



1. Select the Load Combination: DEAD + LIVE.
2. Select PRELIMINARY from Design pull-down menu.

3. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L.
   c. Do not check APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, DL=DECK + SELF WEIGHT, or RE-ANALYZE ALL ADJOINING MEMBERS.
4. Select the connectivity as a HINGE and a ROLLER.
5. Enter an Estimated Self Weight of 30.0 psf and turn ON UPDATE AREA STRUCTURE LOADS.
6. Enter an appropriate Analysis File Name. Preliminary analysis begins.
7. View the shear, moment, deflection, loads and reactions diagrams.

F. Preliminary design of a typical third point steel beam.

1. Select EXECUTE EXCEL in the Excel Data dialog window. The CASM program will become an icon and Excel will be executed loading the steel beam design spreadsheet.
2. Review the steel beam design options listed.
3. Select the W 16 x 26 since it is the lightest possible beam and also the deepest 26 plf beam choice and send the member size to CASM.

>> Note: The actual weight (26.0 plf) of the beam will be used in the quantity take-off calculations. The estimated beam weight (30.0 plf) will still be used for the applied load to the beams, girders and columns. It is possible to revise the beam weight in the floor dead load based on the engineer's judgement. It will not be done in this example since the weights are reasonably close.

4. Return to CASM. The selected beam size will be displayed on both of the third point beams.

>> Note: Since neither third point beam was designed, the selected size was copied to both beams. If both beams were already designed, only the selected beam's size would be updated. Use the Copy Design command to update the other beam's size.

G. Analyze and design the beam parallel to the third point beams on grid line B.

>> Note: Since this beam carries the same loads as the third point beams, the design size and properties only need to be copied from a third point beam.

1. Select COPY DESIGN from the Edit pull-down menu. Handles will appear on all structure designed.
2. Select one of the third point beam handles.
3. Select the beam on grid line B. The beam size will be displayed next to the beam.

4. Double click the right mouse key to end copying the beam design size and properties.

H. Analyze and design the girder perpendicular to the third point beams on grid line 2.

» **Note: Follow similar procedure as used for the third point beam design.**

1. Analyze a Widely Spaced Steel Rolled Section with an estimated self weight of 50.0 plf. Do NOT update the area structure loads. Assume a Hinge and a Roller for connectivity.
2. Design the girder in Excel and select a W 21 x 50, sending the member size to CASM.
3. Return to CASM. The selected girder size will be displayed on the floor plane.

I. Analyze and design the composite metal deck + concrete.

» **Note: The spreadsheet for the design of composite metal deck + concrete is currently under development, therefore the deck size and properties must be manually inserted by the engineer.**

1. Select MODIFY DESIGN from the Edit pull-down menu. Handles will appear on all structural elements drawn.
2. Select the one-way surface handle. The Design dialog window will appear.
3. Select the Material as STEEL.
4. Enter the Description as 2" - 20 GA + 2-1/2" CONC.
5. Enter the Weight of the composite metal deck as 2.0 psf.
6. Enter the pounds per cubic foot Concrete Weight as 145.0 pcf.
7. Enter the pounds per square foot Concrete Weight as 40.0 psf.
8. Enter the Depth as 4.5 inches.
9. Select OK when finished. The selected surface description will be displayed on the floor plane.

J. Perform quantity take-off for the typical interior floor bay.

1. Select QUANTITY TAKE-OFF from the Design pull-down menu. The Quantity Take-off dialog window will appear.
2. Turn ON OVERRIDE CALCULATED SQUARE FOOTAGE OF TOTAL FLOOR AND ROOF PLANES.
3. Enter the Single Bay Dimensions of 24.0 feet by 24.0 feet.
4. Turn ON CURRENT STRUCTURAL PLANE.
5. Enter an appropriate File Name.
6. Select OK to perform the quantity take-off. Notepad will automatically be executed loading quantity take-off output file.

7. Review the quantity take-off.

Scheme B Sample Output:

```
Project     : Bank - Scheme B
Location    : Champaign
Time        : Mon Sep 30, 1991  10:46 AM

**************************** Quantity Take-off ****************************

---------------------------------------------------------------------------
                                 Roof - 2
---------------------------------------------------------------------------
                 Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft

---------------------------------------------------------------------------
                                 Floor - 1
---------------------------------------------------------------------------
                 Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft

STEEL: Widely Spaced Elements
---------------------------------------------------------------------------
                                                      Weight/        Total
Description                           Length Weight  Element   No.  Weight
                                       (ft)   (plf)   (lbs)          (lbs)
---------------------------------------------------------------------------
W 21 x 50                              24.0   50.0   1200.0    1     1200
W 16 x 26                              24.0   26.0    624.0    3     1872
                                       24.0    0.0      0.0    4        0
---------------------------------------------------------------------------
Sum                                                                  3072

Total Weight            :     1.5 tons
Weight Per Square Foot  :     5.3 psf

STEEL: Surface Elements
---------------------------------------------------------------------------
                           Total                Conc    Conc   Total Weight
Description                Depth  Area  Weight Weight  Weight        Conc
                           (in)  (sqft)  (psf)  (pcf)   (psf)  (lbs)  (lbs)
---------------------------------------------------------------------------
2" - 20ga + 2-1/2" Conc.    4.5    576    2.0  145.0    40.0   1152  23040
                            0.0   1152    0.0    0.0     0.0      0      0
---------------------------------------------------------------------------
Sum                                                              1152  23040

Concrete Cubic Yards    :     5.9
Total Weight            :     0.6 tons
```

K. Save the model as QUANT1B.BLD.

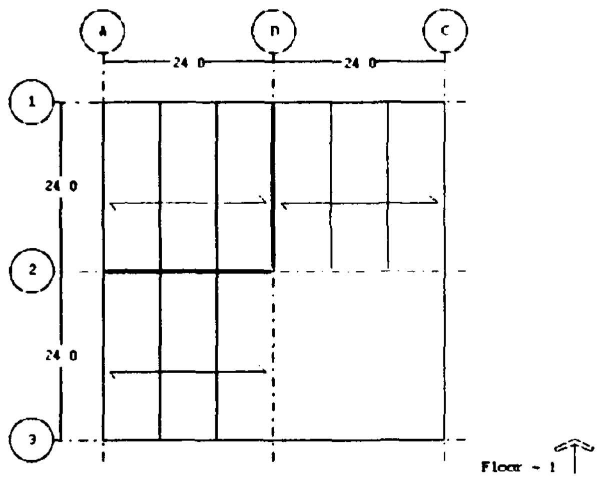
SCHEME C: Steel Composite Beam/Slab

A. Establish Criteria

1. OPEN the file QUANT1.BLD.
2. Change the Project Name to BANK - SCHEME C.

B. Draw Structure

1. The structure is already drawn correctly from Scheme B.



C. Develop Independent Load Cases

1. The floor dead loads are already applied correctly from Scheme B.

D. Establish element parameters to design a typical composite third point steel beam.

1. Select STEEL from the Material pull-down menu.
2. Select COMPOSITE BEAM/SLAB from the Surf/Line pull-down menu.
3. Select either handle in the corner bay A1-B2.
4. Review the Linear Elements and the Element Attributes dialog windows.

E. Preliminary analysis of a typical composite third point steel beam.

1. Select the Load Combination: DEAD + LIVE.
2. Select PRELIMINARY from Design pull-down menu.
3. Select Units options:
   a. Select units of FEET and KIPS.
   b. Verify load combination of D + L.
   c. Turn ON DL=DECK + SELF WEIGHT. Turn OFF APPLY LIVE LOAD REDUCTION, PATTERN OCCUPANCY LIVE LOAD, USE ACTUAL PROPERTIES, and RE-ANALYZE ALL ADJOINING MEMBERS.
4. Select the connectivity as a HINGE and a ROLLER.

5. Enter an Estimated Self Weight of 26.0 psf and turn ON UPDATE AREA STRUCTURE LOADS and turn ON ADD SELF WEIGHT.
6. Enter an appropriate Analysis File Name. Preliminary analysis begins.
7. View the shear, moment, deflection, loads and reactions diagrams.

F. Preliminary design of a typical third point steel beam.

1. Select EXECUTE EXCEL in the Excel Data dialog window. The CASM program will become an icon and Excel will be executed loading the steel beam design spreadsheet.
2. Review the steel composite beam design options listed.
3. Select BEAM CONFIGURATION from the Member pull-down menu and select the Member Type as a BEAM.
4. Select the W 12 x 19 since it is the lightest possible beam and send the member size to CASM.

» Note: The actual weight (19.0 plf) of the beam will be used in the quantity take-off calculations. The estimated beam weight (26.0 plf) will still be used for the applied load to the beams, girders and columns. It is possible to revise the beam weight in the floor dead load based on the engineer's judgement. It will not be done in this example since the weights are reasonably close.

5. Return to CASM. The selected beam size and number of shear studs will be displayed on both of the third point beams.

G. Analyze and design the composite beam parallel to the third point beams on grid line B.

» Note: Since this beam carries the same loads as the third point beams, the design size and properties only need to be copied from a third point beam.

1. Select COPY DESIGN from the Edit pull-down menu. Handles will appear on all structure designed.
2. Select one of the third point beam handles.
3. Select the beam on grid line B. The beam size will be displayed next to the beam.
4. Double click the right mouse key to end copying the beam design size and properties.

H. Analyze and design the girder perpendicular to the third point beams on grid line 2.

» Note: Follow similar procedure as used for the third point beam design.

1. Analyze a Widely Spaced Composite Beam/Slab with an estimated self weight of 40.0 plf. Do NOT update the area structure loads. Assume a Hinge and a Roller for connectivity.
2. Design the girder in Excel.

a. Select BEAM CONFIGURATION from the Member pull-down menu and select the Member Type as GIRDER.

b. Select a W 16 x 40, sending the member size to CASM.

3. Return to CASM. The selected girder size along with the number of shear studs will be displayed on the floor plane.

I. **Analyze and design the composite metal deck + concrete.**

» **Note: The spreadsheet for the design of composite metal deck + concrete is currently under development, therefore the deck size and properties must be manually inserted by the engineer.**

1. Select MODIFY DESIGN from the Edit pull-down menu. Handles will appear on all structural elements drawn.
2. Select the one-way surface handle. The Design dialog window will appear.
3. Select the Material as STEEL.
4. Enter the Description as 2" - 20 GA + 2-1/2" CONC.
5. Enter the Weight of the composite metal deck as 2.0 psf.
6. Enter the pounds per cubic foot Concrete Weight as 145.0 pcf.
7. Enter the pounds per square foot Concrete Weight as 40.0 psf.
8. Enter the Depth as 4.5 inches.
9. Select OK when finished. The selected surface description will be displayed on the floor plane.

J. **Perform quantity take-off for the typical interior floor bay.**

1. Select QUANTITY TAKE-OFF from the Design pull-down menu. The Quantity Take-off dialog window will appear.
2. Turn ON OVERRIDE CALCULATED SQUARE FOOTAGE OF TOTAL FLOOR AND ROOF PLANES.
3. Enter the Single Bay Dimensions of 24.0 feet by 24.0 feet.
4. Turn ON CURRENT STRUCTURAL PLANE.
5. Enter an appropriate File Name.
6. Select OK to perform the quantity take-off. Notepad will automatically be executed loading quantity take-off output file.
7. Review the quantity take-off.

Scheme C Sample Output:

```
Project    : Bank - Scheme C
Location   : Champaign
Time       : Mon Sep 30, 1991  12:56 PM

**************************** Quantity Take-off ****************************
--------------------------------------------------------------------------
                                  Roof - 2
--------------------------------------------------------------------------
                Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft

--------------------------------------------------------------------------
                                 Floor - 1
--------------------------------------------------------------------------
                Plan Area: 24.0 ft x 24.0 ft: 576.0 sqft
```

STEEL: Widely Spaced Elements

Description	Length (ft)	Weight (plf)	Weight/ Element (lbs)	No.	Total Weight (lbs)
W 16 x 40 (58)	24.0	40.0	960.0	1	960
W 12 x 19 (20)	24.0	19.0	456.0	3	1368
	24.0	0.0	0.0	4	0
Sum					2328

```
Total Weight            :   1.2 tons
Weight Per Square Foot  :   4.0 psf
Number of Shear Studs   :   118
```

STEEL: Surface Elements

Description	Total Depth (in)	Area (sqft)	Weight (psf)	Conc Weight (pcf)	Conc Weight (psf)	Total (lbs)	Weight Conc (lbs)
2" - 20 ga + 2-1/2" Conc.	4.5	576	2.0	145.0	40.0	1152	23040
	0.0	1152	0.0	0.0	0.0	0	0
Sum						1152	23040

```
Concrete Cubic Yards    :   5.9
Total Weight            :   0.6 tons
```

K. Save the model as QUANT1C.BLD.

INDEX

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
April 1994

3. REPORT TYPE AND DATES COVERED
Final report

4. TITLE AND SUBTITLE
Tutorial Guide: Computer-Aided Structural Modeling (CASM); Version 5.00

5. FUNDING NUMBERS
Contract No. DACA39-86-C-0024
Work Unit No. AT40-CA-001

6. AUTHOR(S)
David Wickersheimer, Gene McDermott, Ken Taylor, Carl Roth

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Wickersheimer Engineers, Inc.
821 South Neil Street
Champaign, IL 61820

8. PERFORMING ORGANIZATION REPORT NUMBER
Instruction Report ITL-94-1

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
U.S. Army Engineer Waterways Experiment Station, Information Technology Laboratory, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199
U.S. Army Corps of Engineers, Washington, DC 20314-1000

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
Available from National Technical Information Service, 5285 Port Royal Road, Springfield, VA 22161.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The Computer-Aided Structural Modeling (CASM) computer program is designed to aid the structural engineer in the preliminary design and evaluation of structural building systems by the use of three-dimensional (3-D) interactive graphics. CASM allows the structural engineer to quickly evaluate various framing alternatives in order to make more informed decisions in the initial structural evaluation process. The program was developed by the Information Technology Laboratory in conjunction with the Computer Aided Structural Engineering (CASE) Project, Building Systems Task Group.

This release of the CASM is designed to aid the user with design criteria, building loads, and structural framing and design. The various parts of the program are summarized below:

a. Basic design criteria. The user can enter information directly or retrieve information from a user-definable database. The design criteria include information about the project, regional design information, and site-specific design information.

b. Building geometry. The user can assemble the building shape using 3-D primitives (cubes, prisms, spheres, cylinders, etc.) in an easy manner using pull-down menus, icons, and a mouse.

c. Dead and live loads. The user can select and construct dead and live loads from several user-definable menus of building materials and load conditions. These loads can then be applied to any desired area of the building volume.

(Continued)

14. SUBJECT TERMS
Building systems
Computer Aided Structural Engineering (CASE)
Computer programs
Preliminary structural design
Structural modeling
3-Dimensional interactive graphics
3-Dimensional loads

15. NUMBER OF PAGES
303

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED		

13. (Concluded).

d. Snow and wind loads. These loads are automatically calculated in 3-D using information from the basic design criteria database. Wind loads are also calculated for components and cladding and open roof structures. These loads are calculated in accordance with TM 5-809-1.

e. Seismic loads. These loads are calculated based on the equivalent static force method presented in TM 5-809-10.

f. Structural layout. The engineer can easily and rapidly experiment with various framing schemes inside the defined building volume. Beams, girders, joists, girts, columns, walls, and custom trusses are some of the structural elements that can be modeled.

g. Member analysis and preliminary sizing. The user can apply loads to the building geometry from a list of user-defined load cases. The shear, moment, and deflection of selected members may be calculated for various loading conditions (including pattern loads) and connectivity (including continuous beams). The design of a member is performed using a spreadsheet.

Data from the various investigated framing schemes can be edited and printed by CASM and used as justification in a design document.

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Technical Report K-78-1	List of Computer Programs for Computer-Aided Structural Engineering	Feb 1978
Instruction Report O-79-2	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Mar 1979
Technical Report K-80-1	Survey of Bridge-Oriented Design Software	Jan 1980
Technical Report K-80-2	Evaluation of Computer Programs for the Design/Analysis of Highway and Railway Bridges	Jan 1980
Instruction Report K-80-1	User's Guide: Computer Program for Design/Review of Curvilinear Conduits/Culverts (CURCON)	Feb 1980
Instruction Report K-80-3	A Three-Dimensional Finite Element Data Edit Program	Mar 1980
Instruction Report K-80-4	A Three-Dimensional Stability Analysis/Design Program (3DSAD)	
	Report 1: General Geometry Module	Jun 1980
	Report 3: General Analysis Module (CGAM)	Jun 1982
	Report 4: Special-Purpose Modules for Dams (CDAMS)	Aug 1983
Instruction Report K-80-6	Basic User's Guide: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Instruction Report K-80-7	User's Reference Manual: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Technical Report K-80-4	Documentation of Finite Element Analyses	
	Report 1: Longview Outlet Works Conduit	Dec 1980
	Report 2: Anchored Wall Monolith, Bay Springs Lock	Dec 1980
Technical Report K-80-5	Basic Pile Group Behavior	Dec 1980
Instruction Report K-81-2	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CSHTWAL)	
	Report 1: Computational Processes	Feb 1981
	Report 2: Interactive Graphics Options	Mar 1981
Instruction Report K-81-3	Validation Report: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Feb 1981
Instruction Report K-81-4	User's Guide: Computer Program for Design and Analysis of Cast-in-Place Tunnel Linings (NEWTUN)	Mar 1981
Instruction Report K-81-6	User's Guide: Computer Program for Optimum Nonlinear Dynamic Design of Reinforced Concrete Slabs Under Blast Loading (CBARCS)	Mar 1981
Instruction Report K-81-7	User's Guide: Computer Program for Design or Investigation of Orthogonal Culverts (CORTCUL)	Mar 1981
Instruction Report K-81-9	User's Guide: Computer Program for Three-Dimensional Analysis of Building Systems (CTABS80)	Aug 1981
Technical Report K-81-2	Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems	Sep 1981
Instruction Report K-82-6	User's Guide: Computer Program for Analysis of Beam-Column Structures with Nonlinear Supports (CBEAMC)	Jun 1982

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report K-82-7	User's Guide: Computer Program for Bearing Capacity Analysis of Shallow Foundations (CBEAR)	Jun 1982
Instruction Report K-83-1	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Jan 1983
Instruction Report K-83-2	User's Guide: Computer Program for Generation of Engineering Geometry (SKETCH)	Jun 1983
Instruction Report K-83-5	User's Guide: Computer Program to Calculate Shear, Moment, and Thrust (CSMT) from Stress Results of a Two-Dimensional Finite Element Analysis	Jul 1983
Technical Report K-83-1	Basic Pile Group Behavior	Sep 1983
Technical Report K-83-3	Reference Manual: Computer Graphics Program for Generation of Engineering Geometry (SKETCH)	Sep 1983
Technical Report K-83-4	Case Study of Six Major General-Purpose Finite Element Programs	Oct 1983
Instruction Report K-84-2	User's Guide: Computer Program for Optimum Dynamic Design of Nonlinear Metal Plates Under Blast Loading (CSDOOR)	Jan 1984
Instruction Report K-84-7	User's Guide: Computer Program for Determining Induced Stresses and Consolidation Settlements (CSETT)	Aug 1984
Instruction Report K-84-8	Seepage Analysis of Confined Flow Problems by the Method of Fragments (CFRAG)	Sep 1984
Instruction Report K-84-11	User's Guide for Computer Program CGFAG, Concrete General Flexure Analysis with Graphics	Sep 1984
Technical Report K-84-3	Computer-Aided Drafting and Design for Corps Structural Engineers	Oct 1984
Technical Report ATC-86-5	Decision Logic Table Formulation of ACI 318-77, Building Code Requirements for Reinforced Concrete for Automated Constraint Processing, Volumes I and II	Jun 1986
Technical Report ITL-87-2	A Case Committee Study of Finite Element Analysis of Concrete Flat Slabs	Jan 1987
Instruction Report ITL-87-1	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame Structures (CUFRAM)	Apr 1987
Instruction Report ITL-87-2	User's Guide: For Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-83	May 1987
Technical Report ITL-87-6	Finite-Element Method Package for Solving Steady-State Seepage Problems	May 1987
Instruction Report ITL-87-3	User's Guide: A Three Dimensional Stability Analysis/Design Program (3DSAD) Module	Jun 1987
	Report 1: Revision 1: General Geometry	Jun 1987
	Report 2: General Loads Module	Sep 1989
	Report 6: Free-Body Module	Sep 1989

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Instruction Report ITL-87-4	User's Guide: 2-D Frame Analysis Link Program (LINK2D)	Jun 1987
Technical Report ITL-87-4	Finite Element Studies of a Horizontally Framed Miter Gate Report 1: Initial and Refined Finite Element Models (Phases A, B, and C), Volumes I and II Report 2: Simplified Frame Model (Phase D) Report 3: Alternate Configuration Miter Gate Finite Element Studies–Open Section Report 4: Alternate Configuration Miter Gate Finite Element Studies–Closed Sections Report 5: Alternate Configuration Miter Gate Finite Element Studies–Additional Closed Sections Report 6: Elastic Buckling of Girders in Horizontally Framed Miter Gates Report 7: Application and Summary	Aug 1987
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume I, User's Manual	Aug 1987
Instruction Report ITL-87-5	Sliding Stability of Concrete Structures (CSLIDE)	Oct 1987
Instruction Report ITL-87-6	Criteria Specifications for and Validation of a Computer Program for the Design or Investigation of Horizontally Framed Miter Gates (CMITER)	Dec 1987
Technical Report ITL-87-8	Procedure for Static Analysis of Gravity Dams Using the Finite Element Method – Phase 1a	Jan 1988
Instruction Report ITL-88-1	User's Guide: Computer Program for Analysis of Planar Grid Structures (CGRID)	Feb 1988
Technical Report ITL-88-1	Development of Design Formulas for Ribbed Mat Foundations on Expansive Soils	Apr 1988
Technical Report ITL-88-2	User's Guide: Pile Group Graphics Display (CPGG) Post-processor to CPGA Program	Apr 1988
Instruction Report ITL-88-2	User's Guide for Design and Investigation of Horizontally Framed Miter Gates (CMITER)	Jun 1988
Instruction Report ITL-88-4	User's Guide for Revised Computer Program to Calculate Shear, Moment, and Thrust (CSMT)	Sep 1988
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume II, Theory	Feb 1989
Technical Report ITL-89-3	User's Guide: Pile Group Analysis (CPGA) Computer Group	Jul 1989
Technical Report ITL-89-4	CBASIN–Structural Design of Saint Anthony Falls Stilling Basins According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0098	Aug 1989

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Continued)

	Title	Date
Technical Report ITL-89-5	CCHAN–Structural Design of Rectangular Channels According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0097	Aug 1989
Technical Report ITL-89-6	The Response-Spectrum Dynamic Analysis of Gravity Dams Using the Finite Element Method; Phase II	Aug 1989
Contract Report ITL-89-1	State of the Art on Expert Systems Applications in Design, Construction, and Maintenance of Structures	Sep 1989
Instruction Report ITL-90-1	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CWALSHT)	Feb 1990
Technical Report ITL-90-3	Investigation and Design of U-Frame Structures Using Program CUFRBC	May 1990
	Volume A: Program Criteria and Documentation	
	Volume B: User's Guide for Basins	
	Volume C: User's Guide for Channels	
Instruction Report ITL-90-6	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame or W-Frame Structures (CWFRAM)	Sep 1990
Instruction Report ITL-90-2	User's Guide: Pile Group–Concrete Pile Analysis Program (CPGC) Preprocessor to CPGA Program	Jun 1990
Technical Report ITL-91-3	Application of Finite Element, Grid Generation, and Scientific Visualization Techniques to 2-D and 3-D Seepage and Groundwater Modeling	Sep 1990
Instruction Report ITL-91-1	User's Guide: Computer Program for Design and Analysis of Sheet-Pile Walls by Classical Methods (CWALSHT) Including Rowe's Moment Reduction	Oct 1991
Instruction Report ITL-87-2 (Revised)	User's Guide for Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-89	Mar 1992
Technical Report ITL-92-2	Fiinite Element Modeling of Welded Thick Plates for Bonneville Navigation Lock	May 1992
Technical Report ITL-92-4	Introduction to the Computation of Response Spectrum for Earthquake Loading	Jun 1992
Instruction Report ITL-92-3	Concept Design Example, Computer Aided Structural Modeling (CASM)	
	Report 1: Scheme A	Jun 1992
	Report 2: Scheme B	Jun 1992
	Report 3: Scheme C	Jun 1992
Instruction Report ITL-92-4	User's Guide: Computer-Aided Structural Modeling (CASM) - Version 3.00	Apr 1992
Instruction Report ITL-92-5	Tutorial Guide: Computer-Aided Structural Modeling (CASM) - Version 3.00	Apr 1992

(Continued)

WATERWAYS EXPERIMENT STATION REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Concluded)

	Title	Date
Contract Report ITL-92-1	Optimization of Steel Pile Foundations Using Optimality Criteria	Jun 1992
Technical Report ITL-92-7	Refined Stress Analysis of Melvin Price Locks and Dam	Sep 1992
Contract Report ITL-92-2	Knowledge-Based Expert System for Selection and Design of Retaining Structures	Sep 1992
Contract Report ITL-92-3	Evaluation of Thermal and Incremental Co- uction Effects for Monoliths AL-3 and AL-5 of the Melvin Price Locks and Dam	Sep 1992
Instruction Report GL-87-1	User's Guide: UTEXAS3 Slope-Stability Package; Volume IV, User's Manual	Nov 1992
Technical Report ITL-92-11	The Seismic Design of Waterfront Retaining Structures	Nov 1992
Technical Report ITL-92-12	Computer-Aided, Field-Verified Structural Evaluation	
	Report 1: Development of Computer Modeling Techniques for Miter Lock Gates	Nov 1992
	Report 2: Field Test and Analysis Correlation at John Hollis Bankhead Lock and Dam	Dec 1992
	Report 3: Field Test and Analysis Correlation of a Vertically Framed Miter Gate at Emsworth Lock and Dam	Dec 1993
Instruction Report GL-87-1	User's Guide: UTEXAS3 Slope-Stability Package; Volume III, Example Problems	Dec 1992
Technical Report ITL-93-1	Theoretical Manual for Analysis of Arch Dams	Jul 1993
Technical Report ITL-93-2	Steel Structures for Civil Works, General Considerations for Design and Rehabilitation	Aug 1993
Technical Report ITL-93-3	Soil-Structure Interaction Study of Red River Lock and Dam No. 1 Subjected to Sediment Loading	Sep 1993
Instruction Report ITL-93-3	User's Manual—ADAP, Graphics-Based Dam Analysis Program	Aug 1993
Instruction Report ITL-93-4	Load and Resistance Factor Design for Steel Miter Gates	Oct 1993
Technical Report ITL-94-2	User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements	Mar 1994
Instruction Report ITL-94-1	Tutorial Guide: Computer-Aided Structural Modeling (CASM); Version 5.00	Apr 1994
Instruction Report ITL-94-2	User's Guide: Computer-Aided Structural Modeling (CASM); Version 5.00	Apr 1994